GT STRUDL® Version 40

Release Guide

Release Date: November 2021

HEXAGON
PPM

Notice

This GT STRUDL Release Guide is applicable to GT STRUDL Version 40 and later versions for use on PCs under the Microsoft Windows operating systems.

Copyright

U.S. Government Restricted Rights Legend

Intergraph Corporation
305 Intergraph Way
Madison, AL 35758

Documentation

Documentation shall mean, whether in electronic or printed form, User's Guides, Installation Guides, Reference Guides, Administrator's Guides, Customization Guides, Programmer's Guides, Configuration Guides and Help Guides delivered with a particular software product.

Other Documentation

Other Documentation shall mean, whether in electronic or printed form and delivered with software or on Intergraph Smart Support, SharePoint, or box.net, any documentation related to work processes, workflows, and best practices that is provided by Intergraph as guidance for using a software product.

Terms of Use

a. Use of a software product and Documentation is subject to the Software License Agreement ("SLA") delivered with the software product unless the Licensee has a valid signed license for this software product with Intergraph Corporation. If the Licensee has a valid signed license for this software product with Intergraph Corporation, the valid signed license shall take precedence and govern the use of this software product and Documentation. Subject to the terms contained within the applicable license agreement, Intergraph Corporation gives Licensee permission to print a reasonable number of copies of the Documentation as defined in the applicable license agreement and delivered with the software product for Licensee's internal, non-commercial use. The Documentation may not be printed for resale or redistribution.

b. For use of Documentation or Other Documentation where end user does not receive a SLA or does not have a valid license agreement with Intergraph, Intergraph grants the Licensee a non-exclusive license to use the Documentation or Other Documentation for Licensee's internal non-commercial use. Intergraph Corporation gives Licensee permission to print a reasonable number of copies of Other Documentation for Licensee's internal, non-commercial use. The Other Documentation may not be printed for resale or redistribution. This license contained in this subsection b) may be terminated at any time and for any reason by Intergraph Corporation by giving written notice to Licensee.

Disclaimer of Warranties

Except for any express warranties as may be stated in the SLA or separate license or separate terms and conditions, Intergraph Corporation disclaims any and all express or implied warranties including, but not limited to the implied warranties of merchantability and fitness for a particular purpose and nothing stated in, or implied by, this document or its contents shall be considered or deemed a modification or amendment of such disclaimer. Intergraph believes the information in this publication is accurate as of its publication date.

The information and the software discussed in this document are subject to change without notice and are subject to applicable technical product descriptions. Intergraph Corporation is not responsible for any error that may appear in this document.

The software, Documentation and Other Documentation discussed in this document are furnished under a license and may be used or copied only in accordance with the terms of this license. THE USER OF THE SOFTWARE IS EXPECTED TO MAKE THE FINAL EVALUATION AS TO THE USEFULNESS OF THE SOFTWARE IN HIS OWN ENVIRONMENT.

Intergraph is not responsible for the accuracy of delivered data including, but not limited to, catalog, reference and symbol data. Users should verify for themselves that the data is accurate and suitable for their project work.

Limitation of Damages

IN NO EVENT WILL INTERGRAPH CORPORATION BE LIABLE FOR ANY DIRECT, INDIRECT, CONSEQUENTIAL INCIDENTAL, SPECIAL, OR PUNITIVE DAMAGES, INCLUDING BUT NOT LIMITED TO, LOSS OF USE OR PRODUCTION, LOSS OF REVENUE OR PROFIT, LOSS OF DATA, OR CLAIMS OF THIRD PARTIES, EVEN IF INTERGRAPH CORPORATION HAS BEEN ADVISED OF THE POSSIBILITY OF SUCH DAMAGES.

UNDER NO CIRCUMSTANCES SHALL INTERGRAPH CORPORATION'S LIABILITY EXCEED THE AMOUNT THAT INTERGRAPH CORPORATION HAS BEEN PAID BY LICENSEE UNDER THIS AGREEMENT AT THE TIME THE CLAIM IS MADE. EXCEPT WHERE PROHIBITED BY APPLICABLE LAW, NO CLAIM, REGARDLESS OF FORM, ARISING OUT OF OR IN CONNECTION WITH THE SUBJECT MATTER OF THIS DOCUMENT MAY BE BROUGHT BY LICENSEE MORE THAN TWO (2) YEARS AFTER THE EVENT GIVING RISE TO THE CAUSE OF ACTION HAS OCCURRED.

IF UNDER THE LAW RULED APPLICABLE ANY PART OF THIS SECTION IS INVALID, THEN INTERGRAPH LIMITS ITS LIABILITY TO THE MAXIMUM EXTENT ALLOWED BY SAID LAW.

Export Controls

The Software Products and any software products obtained from Intergraph Corporation, its subsidiaries, or distributors, including any technical data related to these products ("Technical Data") are subject to the export control laws and regulations of the United States. Diversion contrary to U.S. law is prohibited. To the extent prohibited by United States or other applicable laws, these Intergraph Corporation software products and any software products obtained from Intergraph Corporation, its subsidiaries or distributors, Technical Data and any derivatives of either, shall not be exported or re-exported, directly or indirectly (including via remote access) under the following circumstances:

a. to Cuba, Iran, North Korea, the Crimean region of Ukraine, or Syria, or any national of these countries or territories.

b. to any person or entity listed on any United States government denial list, including, but not limited to, the United States Department of Commerce Denied Persons, Entities, and Unverified Lists, the United States Department of Treasury Specially Designated Nationals List, and the United States Department of State Debarred List. Visit www.export.gov for more information or follow this link for the screening tool: https://legacy.export.gov/csl-search.

c. to any entity if Customer knows, or has reason to know, the end use of the software product is related to the design, development, production, or use of missiles, chemical, biological, or nuclear weapons, or other un-safeguarded or sensitive nuclear uses.

d. to any entity when Customer knows, or has reason to know, that an illegal reshipment will take place.

Customer shall hold harmless and indemnify PPM for any causes of action, claims, costs, expenses and/or damages resulting to PPM from a breach by Customer or any user of the export compliance restrictions set forth in this Agreement.

Any questions regarding export or re-export of these software products should be addressed to Hexagon PPM, Export Compliance Department, 305 Intergraph Way, Madison, Alabama 35758, USA or at exportcompliance@intergraph.com.

Trademarks

Intergraph®, the Intergraph logo®, Intergraph Smart®, SmartPlant®, SmartMarine®, SmartSketch®, SmartPlant Cloud®, PDS®, FrameWorks®, I-Route, I-Export, Isogen®, SPOOLGEN, SupportManager®, SupportModeler®, SAPPHIRE®, TANK, PV Elite®, CADWorx®, CADWorx DraftPro®, GTSTRUDL®, and CAESAR II® are trademarks or registered trademarks of Intergraph Corporation or its affiliates, parents, subsidiaries. Hexagon and the Hexagon logo are registered trademarks of Hexagon AB or its subsidiaries. Microsoft and Windows are registered trademarks of Microsoft Corporation. ACIS is a registered trademark of SPATIAL TECHNOLOGY, INC. Infragistics, Presentation Layer Framework, ActiveTreeView Ctrl, ProtoViewCtl, ActiveThreed Ctrl, ActiveListBar Ctrl, ActiveSplitter, ActiveToolbars Ctrl, ActiveToolbars Plus Ctrl, and ProtoView are trademarks of Infragistics, Inc. Incorporates portions of 2D DCM, 3D DCM, and HLM by Siemens Product Lifecycle Management Software III (GB) Ltd. All rights reserved. Gigasoft is a registered trademark, and ProEssentials a trademark of Gigasoft, Inc. VideoSoft and VXFlexGrid are either registered trademarks or trademarks of ComponentOne LLC 1991-2017, All rights reserved. Oracle, JD Edwards, PeopleSoft, and Retek are registered trademarks of Oracle Corporation and/or its affiliates. Tribon is a trademark of AVEVA Group plc. Alma and act/cut are trademarks of the Alma company. Other brands and product names are trademarks of their respective owners.

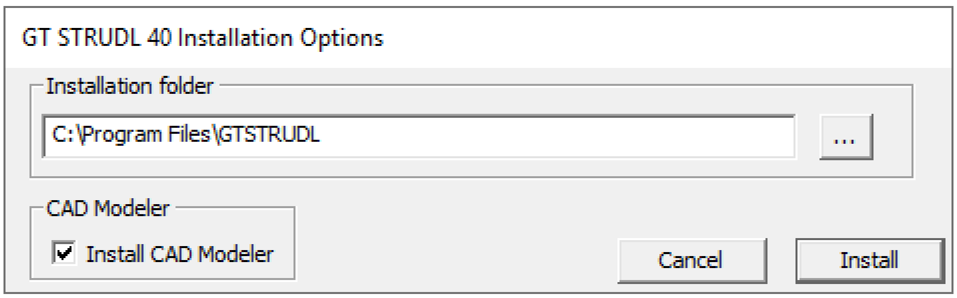
Table of Contents

Chapter 5 Prerelease Features

Chapter 1

Introduction

Version 40 covers GT STRUDL operating on PCs under the Windows 10 x64 based operating system. Installation requires approximately 2.5 GB of storage for GT STRUDL 40. To interactively install GT STRUDL 40, run Setup.exe. Select an installation folder and choose if you wish to install CAD Modeler at this time by checking or clearing the check box. Click Install and choose a language of the Hexagon PPM licensing agreement, and then click the Proceed button when it appears. See the ReadMe.txt file on the CD for information about installing GT STRUDL 40 in batch mode.



CAD Modeler

CAD Modeler is the CAD based structural modeler that gives you the power of AutoCAD® or BricsCAD® to create structural models that can then be passed to GT STRUDL for analysis. If you decide not to install CAD Modeler at this time, you can install it later from <*installation folder*>\40\CADModeler. AutoCAD® or BricsCAD® must be installed on your computer to be able to use CAD Modeler.

Chapter 2 of this release guide presents the new features and enhancements which have been added since the release of Version 2020. Chapter 2 briefly describes an extensive list of new features including the following new features of which a few are highlighted below:

Chapter 3 provides you with details regarding error corrections that have been made since the Version 2020 release.

Chapter 4 describes known problems with Version 40.

Chapter 5 describes prerelease features -- new features which have been developed and subjected to limited testing or features for which the user documentation has not been added to the GT STRUDL User Reference Manual. The command formats and functionality of the prerelease features may change before they become supported features based on additional testing and feedback from users. The Prerelease features are subdivided into Design, Analysis, and General categories.

Chapter 2

New Features in Version 40

This chapter provides you with details regarding new features and enhancements that have been added to many of the functional areas of GT STRUDL in Version 40. This release guide is also available online upon execution of GT STRUDL under menu "Help → Reference Documentation → GT STRUDL Release Guide".

2.1 General

1. SAVE files are now written to a single file, no matter how large the current data base. In addition, the SAVE process is much faster for larger data bases. For example, a GT STRUDL process with 3 GB of internal data took approximately 15 minutes to SAVE in version 2020. In version 40, the SAVE process takes about 10 seconds.

2. GTMenu POINT COORDINATE and LINE INCIDENCES Commands are now in release status.

2.2 Static Analysis

1. Three new MITC class triangular finite elements have been added to the finite element library for Version 40. The element type names, given in the element properties command, are T6CDRL, PBTMITC, and SBTMITC.

   The T6CDRL element is a plane stress element that has nine degrees of freedom, two planar translations and a planar rotation at each joint. The nodal rotation degree of freedom has true coupling with the two translation degrees of freedom. The membrane behavior is derived from an assumed higher order interpolation function for element strains, referred to as ANDES, Assumed Natural Deviatoric Strain.

   The PBTMITC element is a plate bending element that incorporates the three typical plate bending displacement degrees of freedom at each of the three corner nodes. The formulation of PBTMITC is based on Mindlin-Reissner plate theory and incorporates assumed higher order interpolation functions for the strain fields. It is therefore valid for the analysis of thin to thick plates.

   The SBTMITC element has the complete set of six degrees of freedom at each node – u_1, u_2, u_3, θ_1, θ_2, θ_3 - and is formed by the overlay of the T6CDRL plane stress element and the PBTMITC plate bending element. The three-node geometry, accuracy, and stability of these elements make them a versatile choice for all meshing needs.

Documentation:

The TYPE Command for Finite Elements and the GT STRUDL Element Library, Section 2.3.4, Volume 3, GT STRUDL User Reference Manual

The ELEMENT INCIDENCE Command, Section 2.3.5.1, Volume 3, GT STRUDL User Reference Manual

Description of GT STRUDL Finite Elements, Section 2.3.8, Volume 3, GT STRUDL User Reference Manual

2.3 Loadings

1. The CREATE AUTOMATIC LOAD COMBINATION STANDARD command function has been enhanced by the added generation of a load combination descriptor that indicates the specified standard load combination equation. The descriptor is reported when a text reports of applied loading data and load combination/form load analysis results are requested. The following example illustrates a typical CREATE AUTOMATIC LOAD COMBINATION STANDARD command with excerpts from a subsequent PRINT LOAD DATA command for the automatically generated loading combination:

```
CREATE AUTOMATIC COMBO LOADS DESIGN STANDARD ASCE710L Equation 6 PRINT ON
         .
         .
         .
PRINT LOAD DATA LOAD 21 22

 ****************************************
 *  PROBLEM DATA FROM INTERNAL STORAGE  *
 ****************************************

 JOB ID - FR586        JOB TITLE - TX21577 -- Automatic generation of ASCE LRFD load combos

 ACTIVE UNITS -  LENGTH        WEIGHT        ANGLE       TEMPERATURE      TIME
                   FEET          KIP           RAD           DEGF          SEC

          ********** LOADING DATA **********


 LOADING - 21           ASCE710L Eq.6  = 0.9D + 0.9HR + 1.0W                       STATUS - ACTIVE

          COMBINATION GIVEN -  D          0.900      H-         0.900      WX         1.000
```

The descriptor for load combination loading 21 shown above, wherein the character string ASCE710L indicates that the design standard is ASCE 7-10 LRFD, the load combination equation number is 6, and the design load variables “D” represents Dead load, “HR” represents lateral pressure that resists the primary live load, and “W” represents Wind load.

Documentation:

Commands for the Automatic Creation of Standardized Load Combinations, Section 2.1.11.3.6.2, Volume 1, GT STRUDL User Reference Manual

The STORE DESIGN LOAD VARIABLES Command, Section 2.1.11.3.6.3, Volume 1, GT STRUDL User Reference Manual

The CREATE AUTOMATIC LOAD COMBINATION DESIGN Command, Section 2.1.11.3.6.4, Volume 1, GT STRUDL User Reference Manual

2. The WIND AREA command has been added to define areas on a face of a building where the areas are subject to wind loads. There are two types of Wind Face: WALL and ROOF, and the ROOF has three types: GABLE, MONOSLOPE, and MANSARD. The WIND FACE for WALL is defined by Wind Face DIRECTION and Wind Face POSITION AT JOINT while the WIND FACE for ROOF is defined by RIDGE DIRECTION and ROOF PLANE BY JOINT. The WIND FACE definition needs to be specified and the rest is optional. This command basically computes areas on a wind face and distributes them to joints on the face so that the distributed areas will be used to compute joint loads by multiplying the tributary areas by pressures at the joints. Both WIND AREA and WIND PRESSURE names need to be referred under a WIND LOAD ENCLOSED command set to compute joint loads on a wind face. The AREA LIMITS option can be used to exclude openings on a building face as it is used in the AREA LOAD command. The ADDITIONAL BOUNDARY LINES option can be used to enclose more areas by the designated fictitious lines which are specified by two end joints. For example, this can be used to enclose areas along a building base on the ground. An example for the WIND AREA command usage is below:

```
UNITS  FEET  LB    DEG   FAHR  SEC
WIND AREA 'WLZ+'     'Wall Z+'
  WIND FACE                                                     -
    TYPE  WALL                                                  -
    DIRECTION   Z                                               -
    POSITION AT JOINT  169                                      -
    PLANE TOLERANCE        0.167
  ADDITIONAL BOUNDARY LINES                                     -
    TOLERANCE        0.167                                      -
    FROM  169       TO  189
END OF WIND AREA

...

UNITS  FEET  LB    DEG   FAHR  SEC
WIND AREA 'RFZ-'     'Roof Z-'
  WIND FACE                                                     -
    TYPE  ROOF GABLE                                            -
    RIDGE DIRECTION   X                                         -
    ROOF PLANE BY JOINTS  588        568        758             -
    PLANE TOLERANCE        0.167
END OF WIND AREA
```

The PRINT WIND AREA command generates a report of each WIND AREA definition. An example of the report is shown below:

```
{ 4408} > PRINT WIND AREA
```

```
 Wind Area -  'WLZ+'     'Wall Z+'

  --- Wind Area Properties ---

  Wind Face
    Type: Wall
    Direction (Global):  Z
    Position at Joint:  169
    Plane Tolerance:        0.167 [FEET]
    Gross Area:     7200.000 [FEET^2]
  Additional Boundary Lines (by joints)
    Line Tolerance:        0.167 [FEET]
    Line  1:   169        170        171        172        173              -
        174        175        176        177        178        179           -
        180        181        182        183        184        185           -
        186        187        188        189
  Members on Wind Area
    169        170        350        171        351        172              -
    352        173        353        174        354        175              -

...

    1439       1260       1440       1261       1441       1262             -
    1442       1263       1443
  Tributary Areas at Joints [FEET^2]
    Area   1  at Joint  169      :       30.000
    Area   2  at Joint  170      :       60.000

...

    Area  83  at Joint  755      :       60.000
    Area  84  at Joint  756      :       30.000
                 Total Area      :     7200.000 [FEET^2]
  Enclosed Area Count:   60

...

 Wind Area -  'RFZ-'     'Roof Z-'

  --- Wind Area Properties ---

  Wind Face
    Type: Roof Gable
    Direction (Outward Normal): (  0.00000E+00,   0.95783E+00,  -0.28735E+00)
    Roof Plane Defined by 3 Joints:  588        568        758
    Ridge Direction:  X
    Roof Slope Angle for Y-up:   16.70[DEG]
    Roof Slope Angle for Z-up:  106.70[DEG]
    Plane Tolerance:        0.167 [FEET]
    Gross Area:     8352.245 [FEET^2]
  Members on Wind Area
    1264       1265       1266       1267       1268       1269             -
    1270       1271       1272       1273       1274       1275             -

...

    1876       1877       1939       1817       1900       1878             -
    1920       1879       1880       1940
  Tributary Areas at Joints [FEET^2]
    Area   1  at Joint  568      :       26.101
    Area   2  at Joint  569      :       52.202
...
```

```
    Area 104  at Joint  902      :       52.202
    Area 105  at Joint  903      :       52.202
                 Total Area      :     8352.245 [FEET^2]
  Enclosed Area Count:   80
```

Documentation:

The WIND AREA Command, Section 2.1.11.3.9.2.2, Volume 1, GT STRUDL User Reference Manual

The PRINT Commands for Wind Loads, Section 2.1.11.3.9.3, Volume 1, GT STRUDL User Reference Manual

3. The WIND PRESSURE command has been added to build Wind Pressure tables. There are two options to define Wind Pressure: STANDARD and USER-DEFINED. The STANDARD option allows you to set Wind Load parameters based ASCE 7-10 provisions and GT STRUDL will calculate and build the Wind Pressure table for you. On the other hand, if you have your own pressure table, you can directly copy and paste the table into the USER-DEFINED command format so that the table may be used for Wind Load generation. An example for the WIND PRESSURE command usage is below:

```
$
$ Wind Pressures - Standard option: ASCE 7-10
$ Wind Direction: -Z
$

UNITS  FEET  LB    DEG   FAHR  SEC
WIND PRESSURE 'Z-FEPIR1' 'WindDir Z- FullEncl PosIntPres RoofPres1'         -
  STANDARD ASCE7-10
  ELEVATION AXIS  Y
  WIND DIRECTION -Z
  BUILDING DIMENSIONS L       80.000     B      200.000     H       42.000
  SPEED MPH      100.000
  EXPOSURE CATEGORY C
  ENCLOSURE CLASS ENCLOSED  USE POSITIVE INTERNAL PRESSURE
  TOPOGRAPHIC FACTOR KZT        1.000
$  TOPOGRAPHIC FACTOR K1 0.26 K2 0.33 K3 0.14
  DIRECTIONALITY FACTOR        0.850
  GUST FACTOR G        0.850
  USE ROOF PRESSURE CONDITION 1
END OF WIND PRESSURE


...



$
$ Wind Pressures - User-defined option
$

WIND PRESSURE 'UDWPWWZ+' 'User-defined windward wall pressures identical to
Z-FEPIR1' USER-DEFINED
  ELEVATION AXIS  Y
  HEIGHT        0.000  PRESSURE        8.431
  HEIGHT       12.000  PRESSURE        8.431
```

```
  HEIGHT       24.000  PRESSURE        9.738
  HEIGHT       36.000  PRESSURE       10.973
  HEIGHT       39.000  PRESSURE       11.230
  HEIGHT       42.000  PRESSURE       11.471
  HEIGHT       45.000  PRESSURE       11.700
  HEIGHT       48.000  PRESSURE       11.916
END OF WIND PRESSURE


WIND PRESSURE 'UDWPLWZ+' 'User-defined leeward wall pressure identical to Z-
FEPIR1' USER-DEFINED
  ELEVATION AXIS  Y
  HEIGHT       42.000  PRESSURE      -13.880
END OF WIND PRESSURE
```

The PRINT WIND PRESSURE command generates a report of each WIND PRESSURE definition. An example of the report is shown below:

```
{ 4409} > PRINT WIND PRESSURE


UNITS  FEET  LB    DEG   FAHR  SEC

 Wind Pressure -  'Z-FEPIR1' 'WindDir Z- FullEncl PosIntPres RoofPres1'

  Pressure Definition: ASCE 7-10
  Elevation Axis:  Y
  Wind Direction: -Z
  Building Dimensions [ft]
   L:       80.000
   B:      200.000
   h:       42.000
  Wind Speed [mph]:      100.000
  Exposure Category: C
  Topographic Factor Kzt:        1.000
   K1:        0.000
   K2:        0.000
   K3:        0.000
  Directionality Factor Kd:        0.850
  Gust Factor G:        0.850
  Enclosure Class: Enclosed
  Internal Pressure: Positive

  --- Velocity Pressure ---

   Velocity Pressure Exposure Coeff Kz
    Windward:      varying (see table below)
    Leeward :        1.054
    Side    :        1.054
   Velocity Pressure qz [psf]
    Windward:      varying (see table below)
    Leeward :       22.943
    Side    :       22.943

  --- External Pressure ---

   Aspect Ratio L/B:        0.400
   External Pressure Coeff Cp
    Windward:  0.80
    Leeward : -0.50
    Side    : -0.70
   External Pressure pe [psf]
    Windward:      varying (see table below)
    Leeward :       -9.751
    Side    :      -13.651
```

```
  --- Internal Pressure ---

   For Enclosed Building:
    Internal Pressure Coeff GCpi:  0.18
    Internal Pressure pi [psf]:        4.130

  --- Design Wind Pressure (p = pe-pi) ---

   Pressure for Enclosed Building [psf]:
    Leeward :      -13.880
    Side    :      -17.781
    Windward:
             Height [ft]             Kz       qz [psf]       pe [psf]        p [psf]
        1         0.000           0.849        18.472         12.561          8.431
        2        12.000           0.849        18.472         12.561          8.431
        3        24.000           0.937        20.393         13.867          9.738
        4        36.000           1.021        22.210         15.103         10.973
        5        39.000           1.038        22.588         15.360         11.230
        6        42.000           1.054        22.943         15.601         11.471
        7        45.000           1.070        23.278         15.829         11.700
        8        48.000           1.084        23.597         16.046         11.916


...


UNITS  FEET  LB    DEG   FAHR  SEC
 Wind Pressure -  'UDWPWWZ+'                                                   -
    'User-defined windward wall pressures identical to Z-FEPIR1'

  Pressure Definition: User-defined
  Elevation Axis:  Y

  ---  Wind Pressure Input (Sorted)  ---

                Height          Pressure
     1           0.000             8.431
     2          12.000             8.431
     3          24.000             9.738
     4          36.000            10.973
     5          39.000            11.230
     6          42.000            11.471
     7          45.000            11.700
     8          48.000            11.916

  --- Wind Pressure at Joint Heights ---

                Height          Pressure
     1           0.000             8.431
     2          12.000             8.431
     3          24.000             9.738
     4          36.000            10.973
     5          39.000            11.230
     6          42.000            11.471
     7          45.000            11.700
     8          48.000            11.916



UNITS  FEET  LB    DEG   FAHR  SEC
 Wind Pressure -  'UDWPLWZ+'                                                   -
    'User-defined leeward wall pressure identical to Z-FEPIR1'

  Pressure Definition: User-defined
  Elevation Axis:  Y

  ---  Wind Pressure Input (Sorted)  ---

                Height          Pressure
     1          42.000           -13.880
```

```
  --- Wind Pressure at Joint Heights ---

              Height          Pressure
    1          0.000           -13.880
    2         12.000           -13.880
    3         24.000           -13.880
    4         36.000           -13.880
    5         39.000           -13.880
    6         42.000           -13.880
    7         45.000           -13.880
    8         48.000           -13.880
```

Documentation:

The WIND PRESSURE Command, Section 2.1.11.3.9.2.1, Volume 1, GT STRUDL User Reference Manual

The PRINT Commands for Wind Loads, Section 2.1.11.3.9.3, Volume 1, GT STRUDL User Reference Manual

4. The WIND LOAD ENCLOSED command has been added to generate Wind Loads with WIND AREA-WIND PRESSURE associations. A single WIND PRESSURE definition may be associated with multiple WIND AREA definitions since a building or structure has multiple wind faces around it. An example for the WIND LOAD ENCLOSED command usage is below:

```
$
$ Wind Load - (Fully/Partially) Enclosed Structure
$ Wind Direction: -Z
$

UNITS  FEET  LB    DEG   SEC
WIND LOAD ENCLOSED 'Z-FEPIR1' 'WindDir Z- FullEncl PosIntPres RoofPres1'
  WIND PRESSURE 'Z-FEPIR1'  WIND AREA 'WLZ+'
  WIND PRESSURE 'Z-FEPIR1'  WIND AREA 'WLZ-'
  WIND PRESSURE 'Z-FEPIR1'  WIND AREA 'WLX+'
  WIND PRESSURE 'Z-FEPIR1'  WIND AREA 'WLX-'
  WIND PRESSURE 'Z-FEPIR1'  WIND AREA 'RFZ+'
  WIND PRESSURE 'Z-FEPIR1'  WIND AREA 'RFZ-'
END OF WIND LOAD ENCLOSED

...

$
$ Wind Load - (Fully/Partially) Enclosed Structure
$ Wind Direction: -X
$

UNITS  FEET  LB    DEG   SEC
WIND LOAD ENCLOSED 'X-FEPIR1' 'WindDir X- FullEncl PosIntPres RoofPres1'
  WIND PRESSURE 'X-FEPIR1'  WIND AREA 'WLZ+'
  WIND PRESSURE 'X-FEPIR1'  WIND AREA 'WLZ-'
  WIND PRESSURE 'X-FEPIR1'  WIND AREA 'WLX+'
  WIND PRESSURE 'X-FEPIR1'  WIND AREA 'WLX-'
  WIND PRESSURE 'X-FEPIR1'  WIND AREA 'RFZ+'
  WIND PRESSURE 'X-FEPIR1'  WIND AREA 'RFZ-'
END OF WIND LOAD ENCLOSED
```

The PRINT WIND LOAD command generates a report of each WIND LOAD definition. An example of the report is shown below:

```
{ 4410} > PRINT WIND LOAD


/=====================================================================================================/
/=====================================================================================================/

 Loading                                Description
---------     ----------------------------------------------------------------------

 Z-FEPIR1      WindDir Z- FullEncl PosIntPres RoofPres1

                Wind Pressure   Pressure Def.     Wind Area     Wind Face Type   Wind Face Direction
              /---------------/---------------/---------------/----------------/---------------------/
                  Z-FEPIR1        ASCE 7-10       WLZ+              Wall          Z     Windward

                                                  WLZ-              Wall         -Z     Leeward

                                                  WLX+              Wall          X     Side

                                                  WLX-              Wall         -X     Side

                                                  RFZ+              Roof         Windward

                                                  RFZ-              Roof         Leeward



 Wind Pressure-Area Pair        Joint                  Wind Load Parameters for Wall
-------------------------     -----------     -------------------------------------------------------
   Z-FEPIR1 - WLZ+
                                               Wind Face Type:                 Wall
                                                         Direction:             Z   Windward

                                               Use PRINT WIND PRESSURE command to see more details.

                               169
                                               Joint Position in Y [ft]:                              0.000
                                               Tributary Area at Joint [ft^2]:                       30.000
                                               Design Wind Pressure p at Joint [psf]:                 8.431
                                               Joint Wind Force WFZ [lb]:                                 -
252.932
                               170
                                               Joint Position in Y [ft]:                              0.000
                                               Tributary Area at Joint [ft^2]:                       60.000
                                               Design Wind Pressure p at Joint [psf]:                 8.431
                                               Joint Wind Force WFZ [lb]:                                 -
505.864
...
                               755
                                               Joint Position in Y [ft]:                             36.000
                                               Tributary Area at Joint [ft^2]:                       60.000
                                               Design Wind Pressure p at Joint [psf]:                10.973
                                               Joint Wind Force WFZ [lb]:                                 -
658.393
                               756
                                               Joint Position in Y [ft]:                             36.000
                                               Tributary Area at Joint [ft^2]:                       30.000
                                               Design Wind Pressure p at Joint [psf]:                10.973
                                               Joint Wind Force WFZ [lb]:                                 -
329.196




...




 Wind Pressure-Area Pair        Joint                  Wind Load Parameters for Roof
-------------------------     -----------     -------------------------------------------------------
   Z-FEPIR1 - RFZ+
                                               Wind Face Type:                 Roof
                                               Roof Type:                      Gable
                                                    Slope Angle [Deg]:               16.699
                                                    Area (Gross) [ft^2]:           8352.245
```

```
                                                     Area (Actual) [ft^2]:          8352.245
                                                     Outward Normal:          (  0.00000E+00,   0.95783E+00,
0.28735E+00)
                                                Ridge Direction:                 X
                                                Wind Direction:                 -Z
                                                Mean Roof Height h [ft]:              42.000
                                                Building Length L [ft]:               80.000
                                                Aspect Ratio h/L:                      0.525
                                                Velocity Pressure qh [psf]:           22.943
                                                Gust Factor G:                         0.850
                                                Internal Pressure Sign:         Positive (+)

                                                Roof Pressure Table: Wind Direction Normal to Ridge
                                                     Face Direction:            Windward
                                                     Pressure Condition:        1
                                                     Pressure Coeff. Cp (uniform over roof):               -
0.613
                                                External Pressure pe (uniform over roof) [psf]:            -
11.955
                                                Internal   Pressure   pi   (uniform   over   roof)   [psf]:
4.130
                                                Design Wind Pressure p (uniform over roof) [psf]:          -
16.085
                              736
                                                Tributary Area at Joint [ft^2]:                       26.101
                                                Joint Wind Force [lb]:
                                                      WFX:                                             0.000
                                                      WFY:                                           402.123
                                                      WFZ:                                           120.637
                              737
                                                Tributary Area at Joint [ft^2]:                       52.202
                                                Joint Wind Force [lb]:
                                                      WFX:                                             0.000
                                                      WFY:                                           804.246
                                                      WFZ:                                           241.274
...

                              839
                                                Tributary Area at Joint [ft^2]:                       52.202
                                                Joint Wind Force [lb]:
                                                      WFX:                                             0.000
                                                      WFY:                                           804.246
                                                      WFZ:                                           241.274
                              840
                                                Tributary Area at Joint [ft^2]:                       52.202
                                                Joint Wind Force [lb]:
                                                      WFX:                                             0.000
                                                      WFY:                                           804.246
                                                      WFZ:                                           241.274



...


/=======================================================================================================/
/=======================================================================================================/

 Loading                                 Description
---------     ------------------------------------------------------------------------

 X-FEPIR1      WindDir X- FullEncl PosIntPres RoofPres1

                Wind Pressure   Pressure Def.     Wind Area     Wind Face Type   Wind Face Direction
              /---------------/---------------/---------------/----------------/---------------------/
                  X-FEPIR1        ASCE 7-10       WLZ+              Wall          Z     Side

                                                  WLZ-              Wall         -Z     Side

                                                  WLX+              Wall          X     Windward

                                                  WLX-              Wall         -X     Leeward

                                                  RFZ+              Roof         Parallel

                                                  RFZ-              Roof         Parallel



 Wind Pressure-Area Pair        Joint                  Wind Load Parameters for Roof
-------------------------     -----------     --------------------------------------------------------
   X-FEPIR1 - RFZ+
                                                Wind Face Type:                 Roof
                                                Roof Type:                      Gable
                                                     Slope Angle [Deg]:               16.699
```

```
                                                    Area (Gross) [ft^2]:          8352.245
                                                    Area (Actual) [ft^2]:         8352.245
                                                    Outward Normal:         (  0.00000E+00,   0.95783E+00,
0.28735E+00)
                                               Ridge Direction:                X
                                               Wind Direction:                -X
                                               Mean Roof Height h [ft]:             42.000
                                               Building Length L [ft]:             200.000
                                               Aspect Ratio h/L:                     0.210
                                               Velocity Pressure qh [psf]:          22.943
                                               Gust Factor G:                        0.850
                                               Internal Pressure Sign:        Positive (+)

                                               Roof Pressure Table: Wind Direction Parallel to Ridge
                                                    Face Direction:           Parallel
                                                    Pressure Condition:       1
                                                    Windward Edge Position in X [ft]:             200.000
                                               Internal   Pressure   pi   (uniform   over   roof)   [psf]:
4.130
                              736
                                               Joint Position in X [ft]:                            0.000
                                               Horizontal    Distance    from    Windward    Edge    [ft]:
200.000
                                               Roof Pressure Coeff. Cp at Joint:                        -
0.300
                                               External Pressure pe at Joint [psf]:                     -
5.850
                                               Design Wind Pressure p [psf]:                            -
9.980
                                               Tributary Area at Joint [ft^2]:                     26.101
                                               Joint Wind Force [lb]:
                                                     WFX:                                           0.000
                                                     WFY:                                         249.503
                                                     WFZ:                                          74.851
                              737
                                               Joint Position in X [ft]:                           10.000
                                               Horizontal    Distance    from    Windward    Edge    [ft]:
190.000
                                               Roof Pressure Coeff. Cp at Joint:                        -
0.300
                                               External Pressure pe at Joint [psf]:                     -
5.850
                                               Design Wind Pressure p [psf]:                            -
9.980
                                               Tributary Area at Joint [ft^2]:                     52.202
                                               Joint Wind Force [lb]:
                                                     WFX:                                           0.000
                                                     WFY:                                         499.005
                                                     WFZ:                                         149.702

...

                              839
                                               Joint Position in X [ft]:                            0.000
                                               Horizontal    Distance    from    Windward    Edge    [ft]:
200.000
                                               Roof Pressure Coeff. Cp at Joint:                        -
0.300
                                               External Pressure pe at Joint [psf]:                     -
5.850
                                               Design Wind Pressure p [psf]:                            -
9.980
                                               Tributary Area at Joint [ft^2]:                     52.202
                                               Joint Wind Force [lb]:
                                                     WFX:                                           0.000
                                                     WFY:                                         499.005
                                                     WFZ:                                         149.702
                              840
                                               Joint Position in X [ft]:                            0.000
                                               Horizontal    Distance    from    Windward    Edge    [ft]:
200.000
                                               Roof Pressure Coeff. Cp at Joint:                        -
0.300
                                               External Pressure pe at Joint [psf]:                     -
5.850
                                               Design Wind Pressure p [psf]:                            -
9.980
                                               Tributary Area at Joint [ft^2]:                     52.202
                                               Joint Wind Force [lb]:
                                                     WFX:                                           0.000
                                                     WFY:                                         499.005
                                                     WFZ:                                         149.702
```

Documentation:

The WIND LOAD ENCLOSED Command, Section 2.1.11.3.9.2.3, Volume 1, GT STRUDL User Reference Manual

The PRINT Commands for Wind Loads, Section 2.1.11.3.9.3, Volume 1, GT STRUDL User Reference Manual

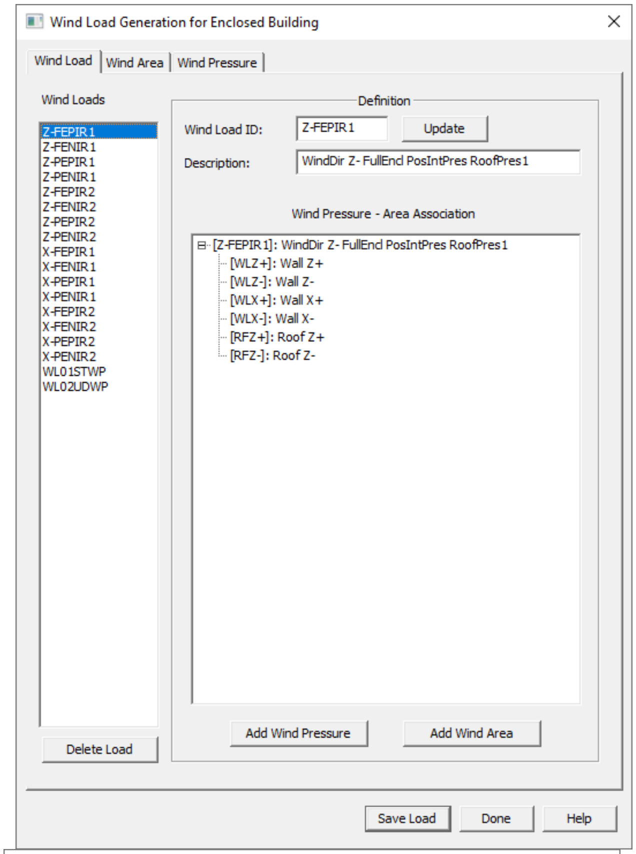
5. The option ‘ASCE7-16’ has been added to the WIND LOAD OPEN, SEISMIC LOADING and CREATE AUTOMATIC LOAD COMBINATIONS DESIGN commands. See the documentation in Volume 1 of the Reference Manual for details.

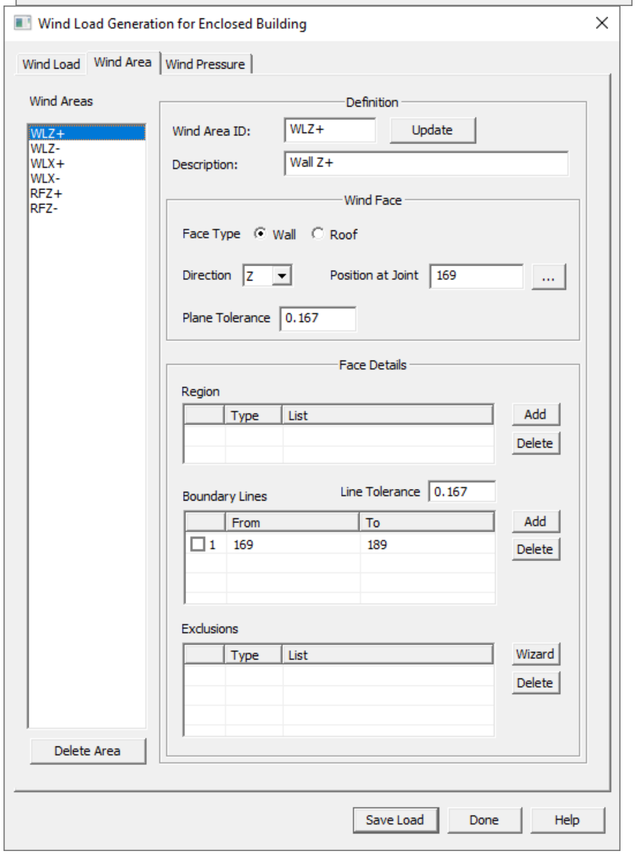
Documentation:

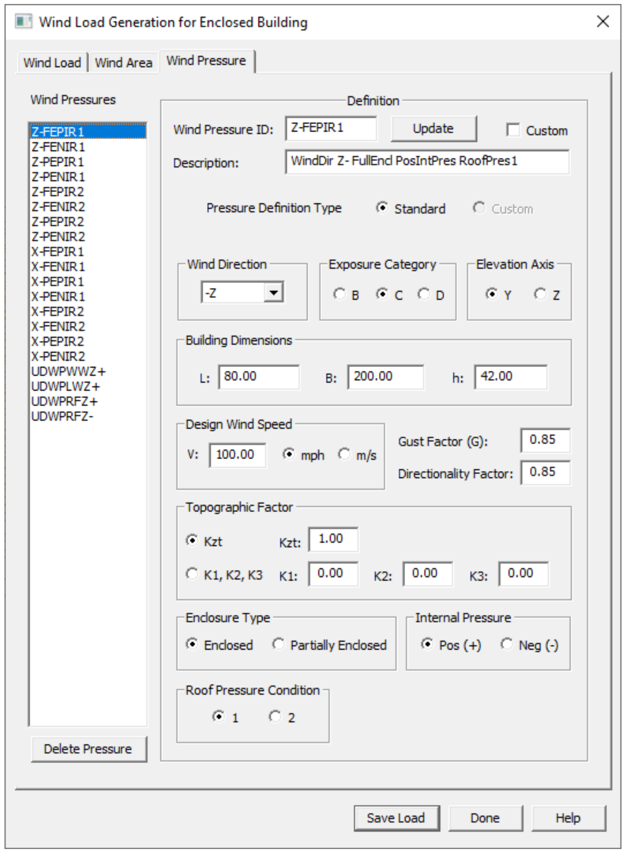
2.1.11.3.6.4 The CREATE AUTOMATIC LOAD COMBINATIONS DESIGN Command.
2.1.11.3.9.1 The WIND LOAD Command
2.1.11.3.10.1 The SEISMIC LOAD Command

2.4 GT STRUDL Output Window (GTShell)

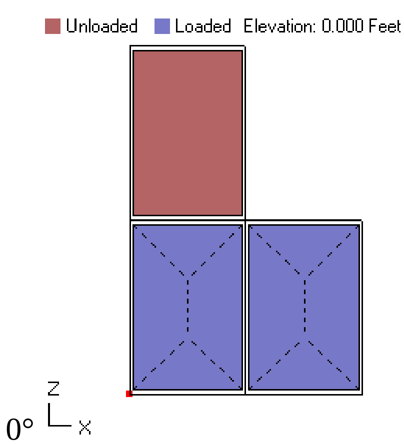
1. The “Wind Loads for Enclosed Building” dialog has been added to GTShell to generate the following new commands for this release: WIND AREA, WIND PRESSURE, and WIND LOAD ENCLOSED. The dialog has three tabs of Wind Load, Wind Area, and Wind Pressure as the commands are separated. The separation makes the user follow steps with the tabs for wind load generation. The Wind Area tab allows the user to include or exclude areas on a wind face of a building. The Wind Pressure tab allows the user to generate height-pressure tables based on either the ASCE 7-10 standard or custom height-pressure table. Once the Wind Area and Wind Pressure are defined, the user may generate Wind Loads on the Wind Load tab by associating the defined Wind Areas and Wind Pressures. As the last step, the user may click on the “Save Load” button to generate all the commands and execute them.



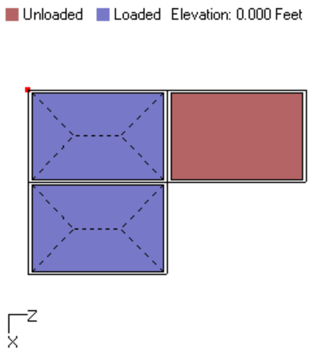




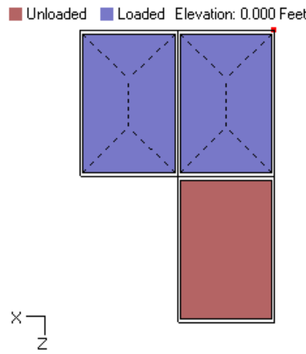
2. The Area Load dialog has had two new buttons added: Rotate Right and Rotate Left. These buttons rotate the area view 90 degrees clockwise (Right) or counterclockwise (Left). The new buttons will also appear when the Wizard view is called from GTMenu Area Load or Wind Load.



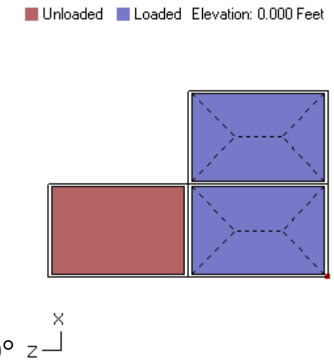
Subsequent Rotate Right:



90°



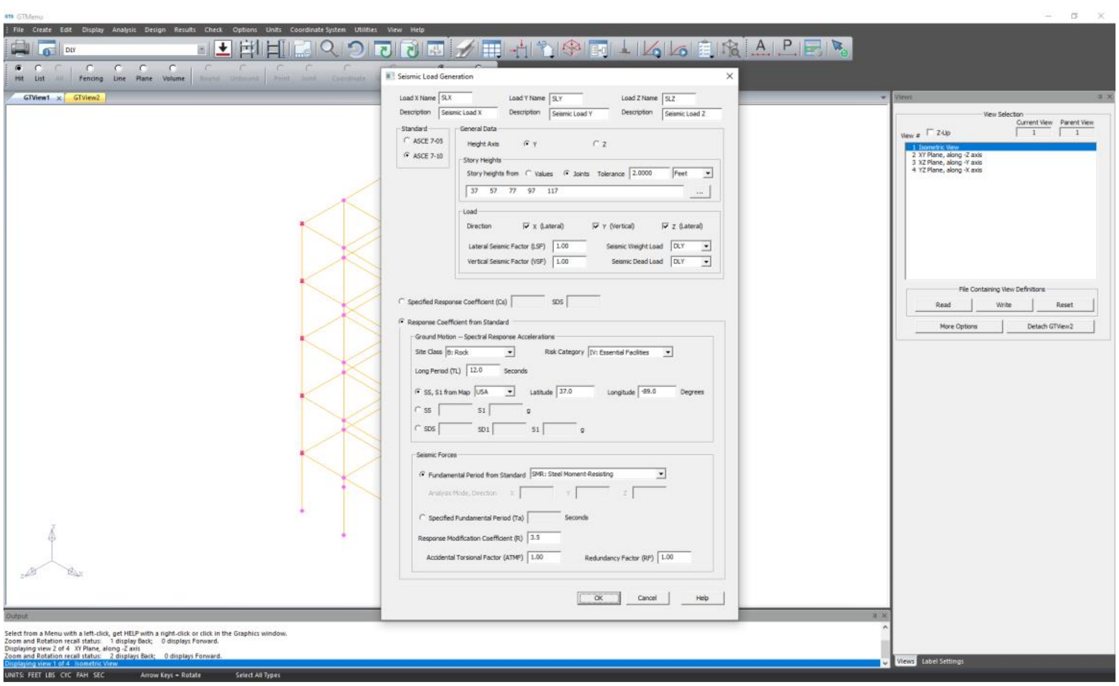
180°

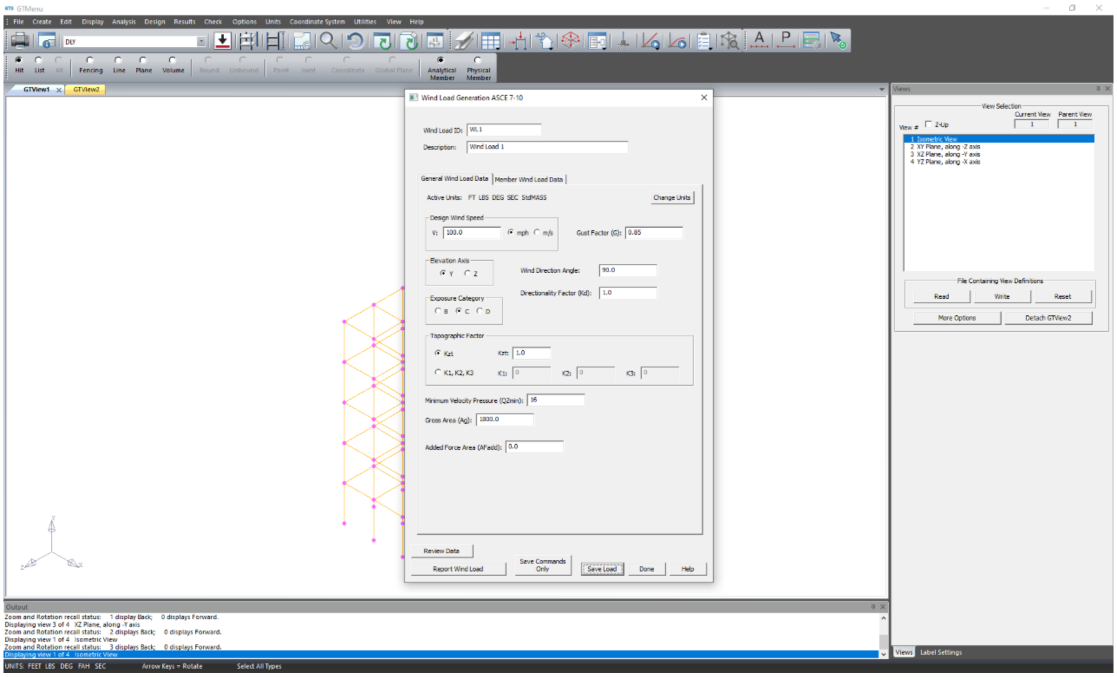


270°

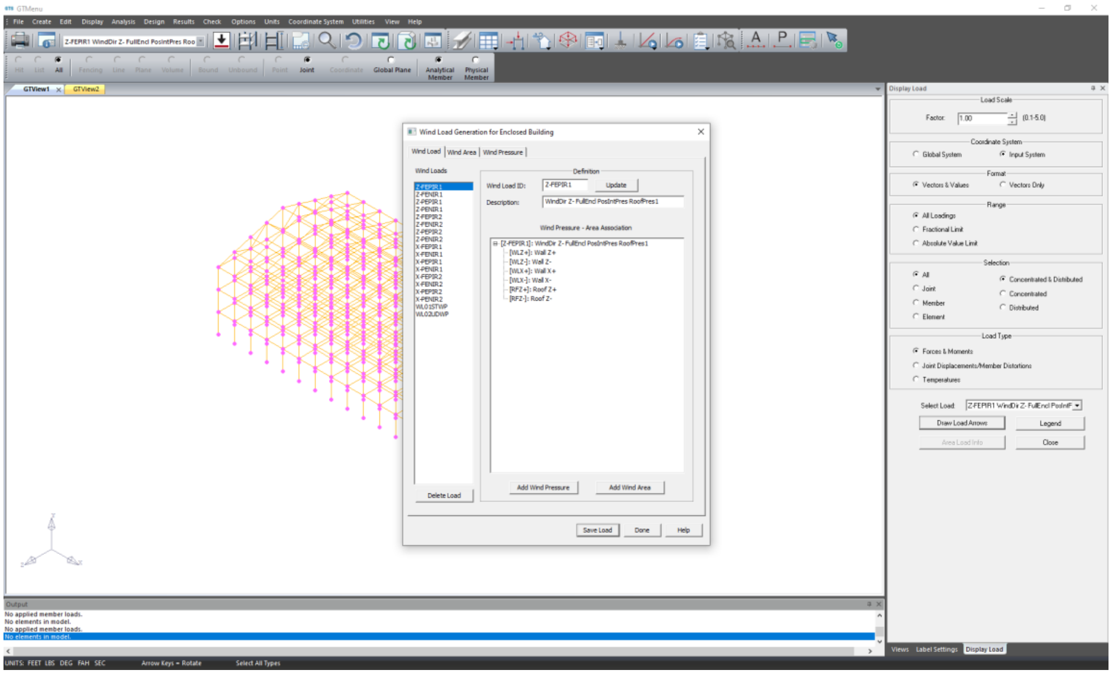
2.5 GTMenu

1. The GTShell dialogs for "Seismic Load Generation" and "Wind Load Generation ASCE 7-05/10" have been added identically to GTMenu so that the user does not need to exit GTMenu to create Seismic Loads or Wind Loads (open structures). The user can open the dialogs at the menu: Create → Loads → Seismic/Wind Loads (Open).

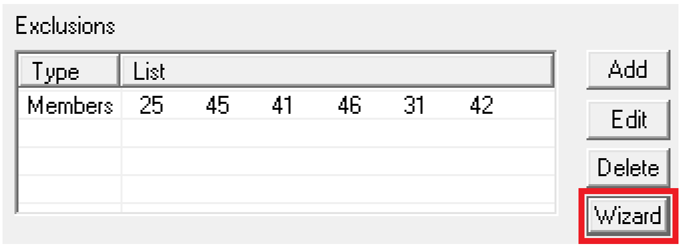




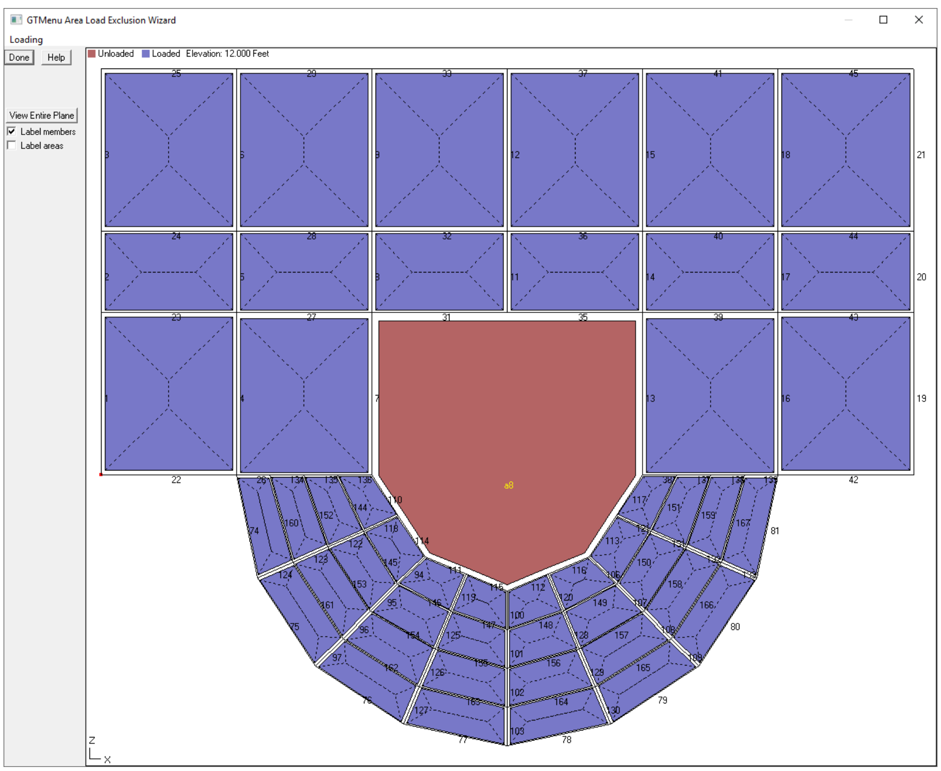
2. The new GTShell dialog “Wind Load Generation for Enclosed Building” for this release has been added identically to GTMenu so that the user does not need to exit GTMenu to create Wind Loads for enclosed structures. The dialog is opened using the menu: Create → Loads → Wind Loads (Enclosed).



3. A new option has been added to the Area Load dialog, the Exclusion Wizard. This new wizard makes selecting regions within the Area Load to be excluded (not loaded) much easier and graphically oriented. The new wizard is accessed from the Exclusion detail with a new “Wizard” button:



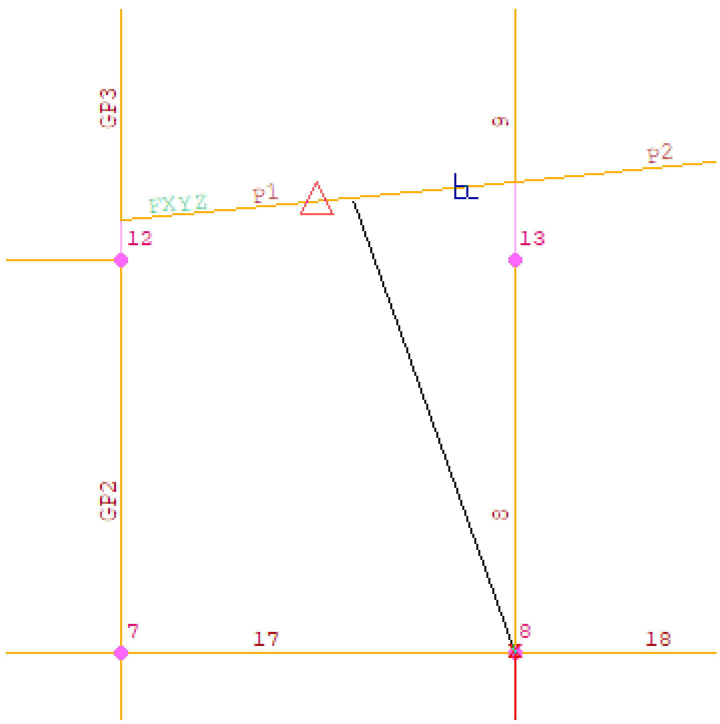
The same wizard used in the Shell will open, allowing the user to select the sub-areas within the current Elevation to be loaded or excluded by picking sub-areas.



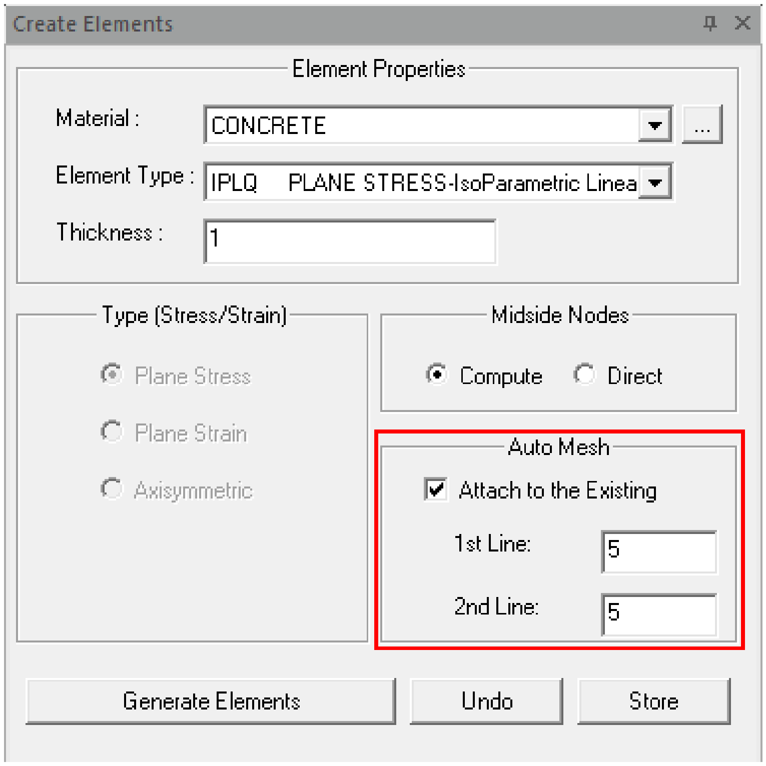
Blue sub-areas will be loaded, and red sub-areas will be excluded. Click the Done button to return to the Area Load dialog and the changes will be inserted into the Exclusions list. Exclusions from the Wizard are always specified as a Members list of members that lie on the perimeter of the excluded sub-area, even if a different type (Coordinates or Joints list) was specified before using the Wizard. The Wizard can be used as many times as necessary for an Elevation.

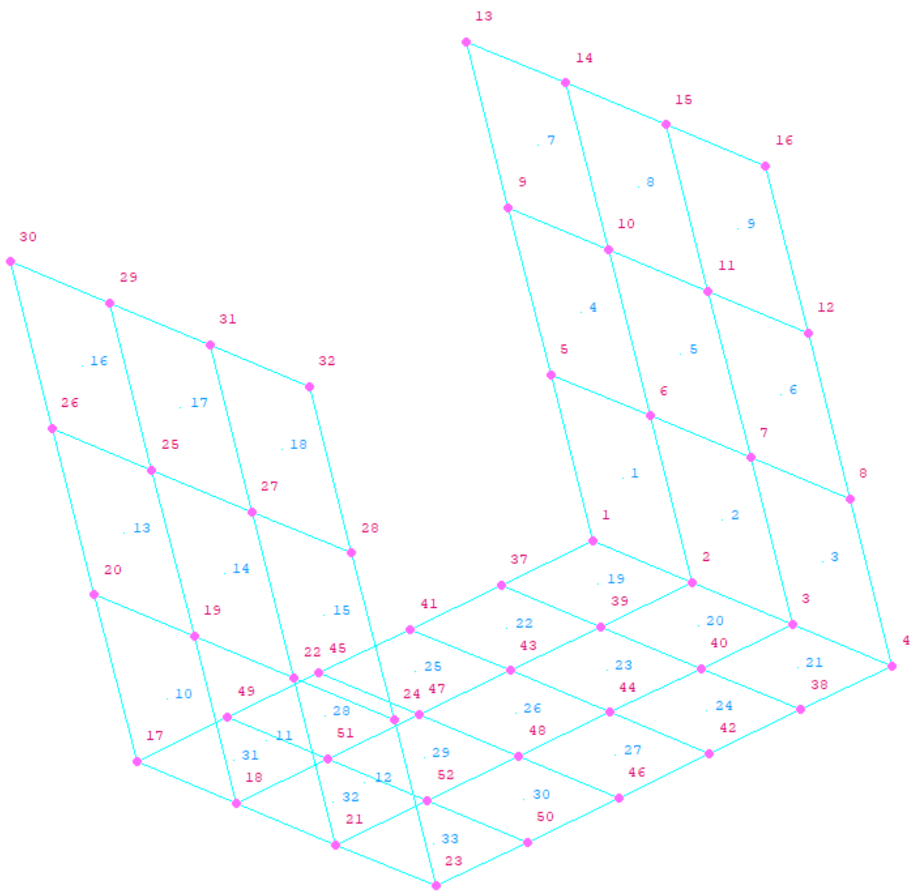
The Exclusion Wizard is available for both Create and Edit Area Loads.

4. Snap options to existing members have been added to the Place Members dialog. These are active for "Connect End to End" and "Specify Start & End" Definition Methods of the dialog. Mid-point snap and/or perpendicular snap options to an existing member are available as shown in figure below. The snap options are added to analytical and physical members. If the mouse cursor is on an analytical or physical member, a triangle for the mid-point snap and/or a right-angle for the perpendicular snap to the member will be displayed. When a snap symbol is clicked, a member will be placed from the last selected point to the snap corresponding to the selected symbol.

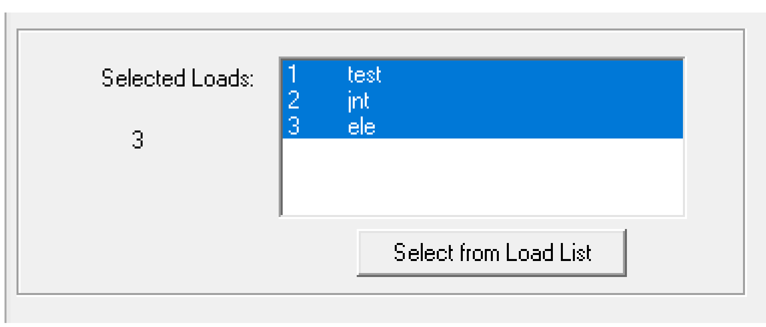


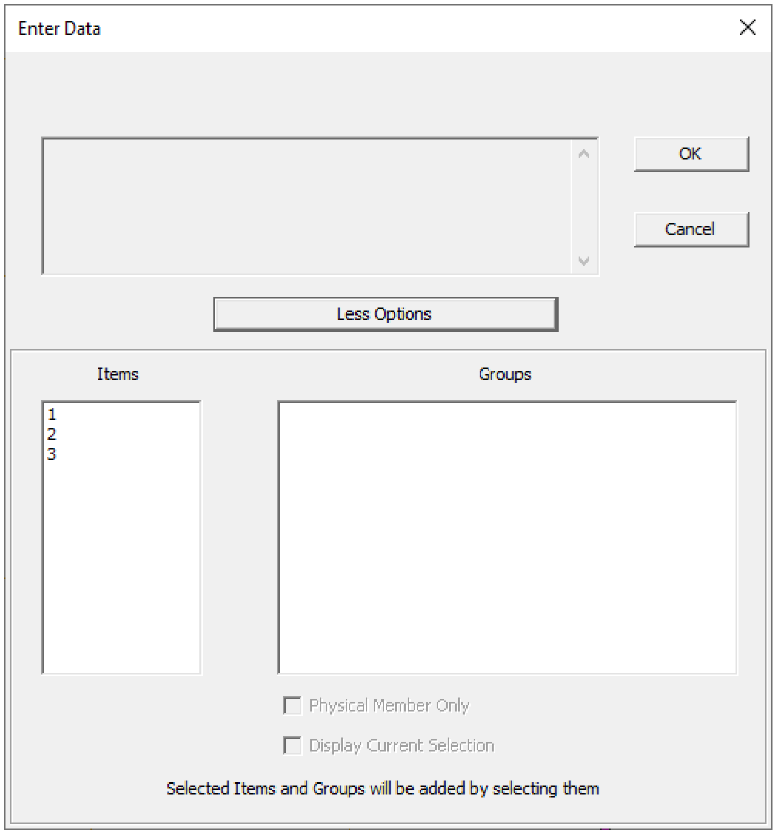
5. Auto finite element mesh capabilities have been added to the Create Elements dialog. The auto mesh capabilities are only available for 4 nodes square element. When a 4 nodes element is selected along clockwise or counterclockwise from the Element Type, the auto mesh is enabled. 1st line from the dialog is composed of 1st and 2nd selected joints and 2nd line is composed of 2nd and 3rd selected joints. The number of discretized elements from the auto finite element mesh can be determined by user-defined numbers at the edit box for each line and/or the number of elements automatically detected along each determined line. In the following figure, 1st line and 2nd line are set to the 5 elements along each line. As shown in below finite element figure, joints 1, 4, 23, and 17 are selected to create finite elements. the number of the user-defined elements for the 1st and 2nd lines are 5 elements. Auto detected elements along 1st line are 3 elements which are applied to the finite elements mesh. For 2nd line, 5 elements are applied to the finite elements mesh because there is no detected element.



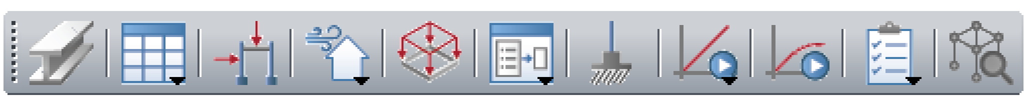


6. Multiple selections of loads for envelop have been added at Member Forces dialog and Finite Element Stress Contour dialog. Using Ctrl key, single load can be selected and deselected. Using Shift key, box selection and deselection are available. In addition, by clicking “Select from Load List” button, load list option in the following figure can be used.

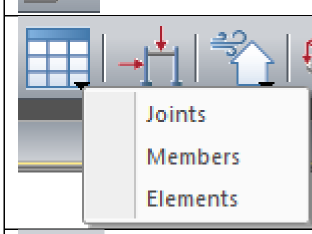
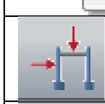
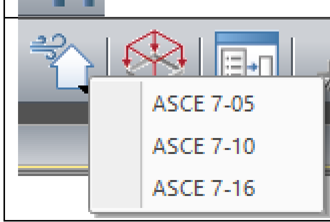


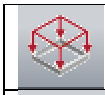
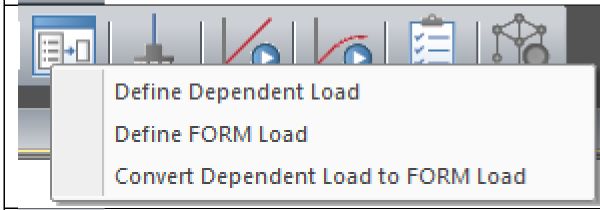
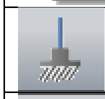
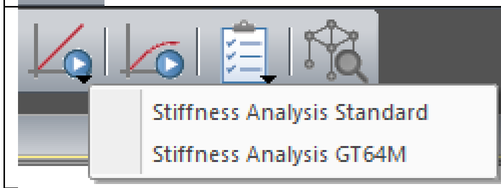
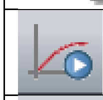
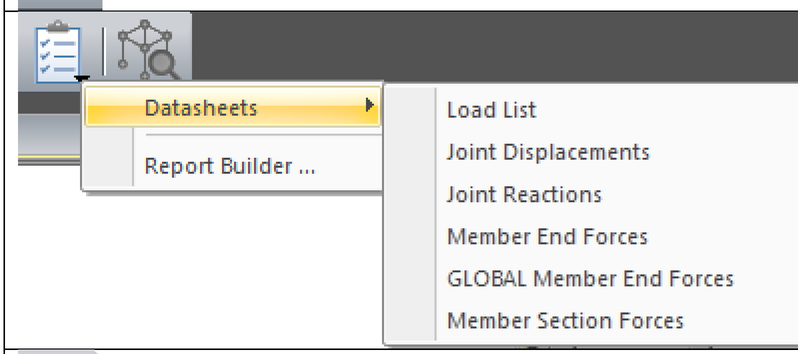
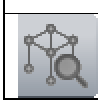


7. A new toolbar to allow users to perform routine actions has been added in GTMenu.

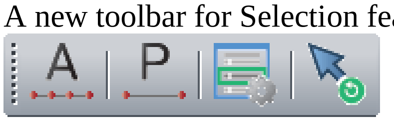


where:

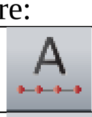
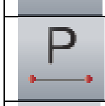
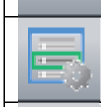
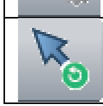
	Place Member Dialog
	Model Datasheet
	Create Load Dialog
	Wind Load Dialog

	Area Load Dialog
	• Dependent Load • Form Load
	Create Support Dialog
	Stiffness Analysis
	Nonlinear Analysis
	Check Results
	Steel Design Dialog

8. A new toolbar for Selection features has been added in GTMenu



where:

	Analytical Member
	Physical Member
	List option for Selection Sets
	Reset current Selection Sets

9. The user can now maintain persistence Selection Sets for joints, members, and elements. These appropriate Selection Set is used when an action is invoked in a dialog.

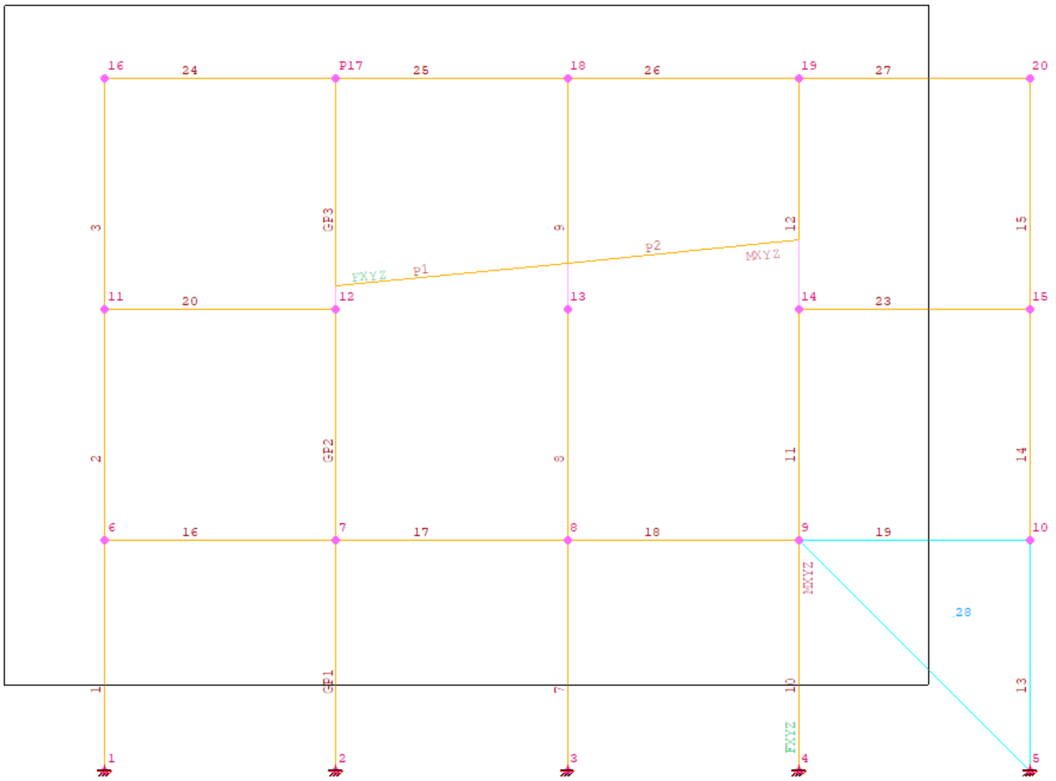
Objects can be added to the Selection Sets as follows:

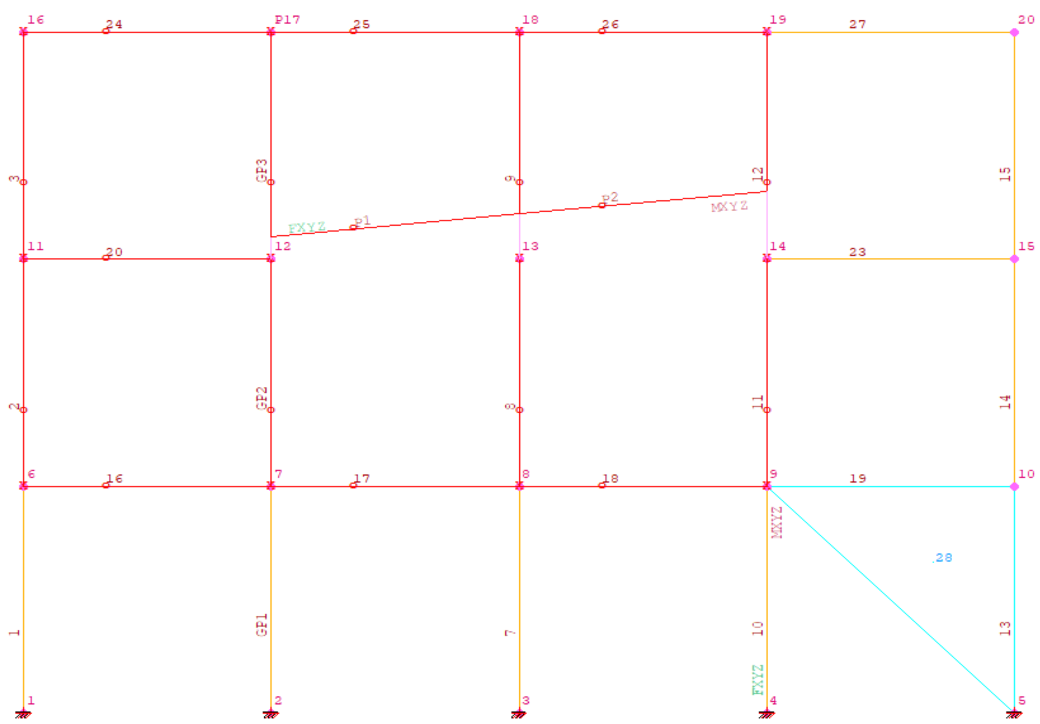
- Window or box selection from left to right will select joints, members, and elements which are completely within the box.
- Window or box selection from right to left will select joints, members, and elements which are partially within the box.
- Single joint, member, and element picking will select only the joint, member, and element.

Objects are removed from the Selection Sets by pressing the Ctrl key while using any of the selection methods above. Additionally, the Esc key will reset the Selection Sets based on selection types.

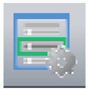
If an action is invoked in a dialog and the corresponding Selection Set is empty, the user will be prompted to select the objects.

The following figures show an example of a window selection from left to right. And the selected objects.

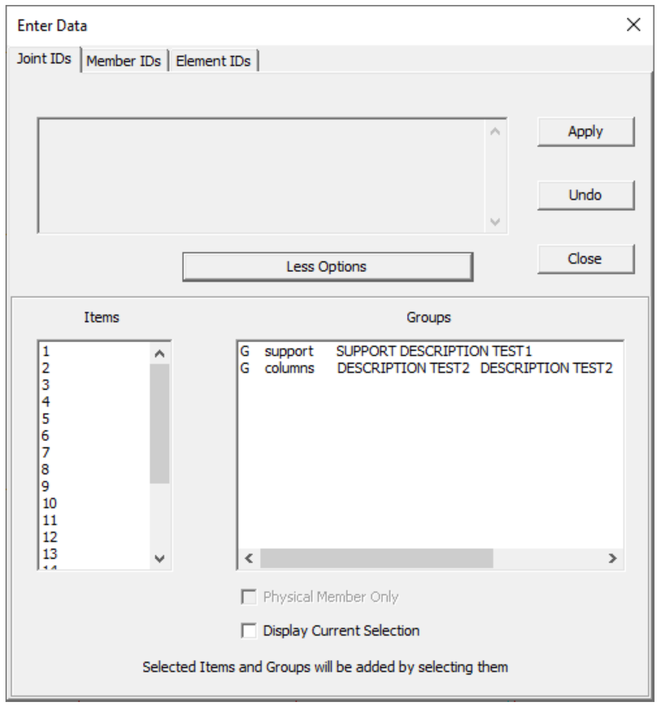

16
24
P17
25
18
26
19
27
20
3
GP3
9
12
P2
MXYZ
FXYZ
P1
15
11
20
12
13
14
23
15
2
GP2
8
11
14
6
16
7
17
8
18
9
19
10
MXYZ
28
1
GP1
7
10
13
FXYZ
1
2
3
4
5


16
24
P17
25
18
26
19
27
20
3
GP3
9
12
P2
MXYZ
FXYZ
P1
15
11
20
12
13
14
23
15
2
GP2
8
11
14
6
16
7
17
8
18
9
19
10
MXYZ
28
1
GP1
7
10
13
FXYZ
1
2
3
4
5

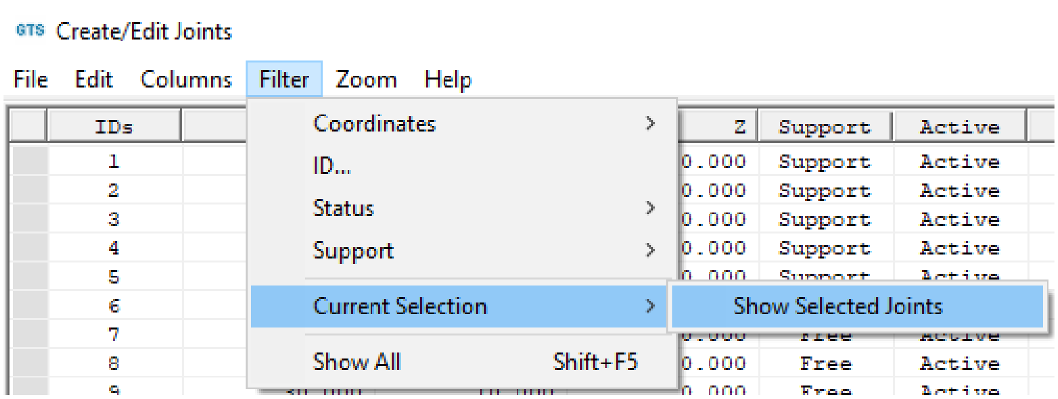
As an alternative to selecting objects in the screen, the user can use the list option to update the Selection Sets. This option can be invoked from the selection toolbar

 : List option for new selection

Existing joint, member, element Selection Sets are displayed and highlighted when using this option. After editing the list of joints, members, and/or elements, the Apply button will update the Selection Sets.

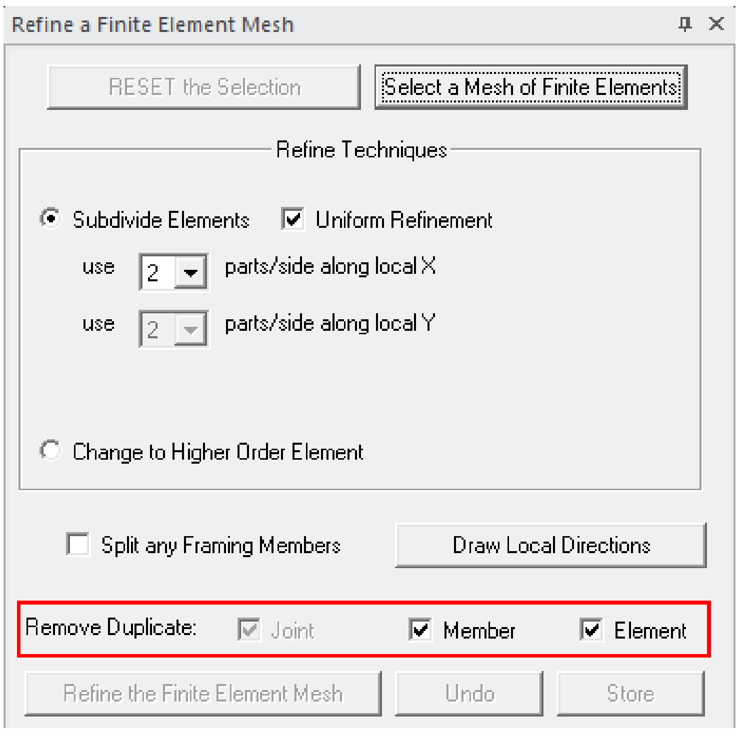


10. Options to filter joints, members, and elements based on the current selection set have been added to Model Joint, Model Member, and Model Element datasheets. An example of how to filter Joints is shown below.



Also, the Results datasheets (Joint Displacement, Joint Reaction, Member End Force, Member Section Force, Global Member End Force, and Code Check) have been updated with these filter options.

11. Options to remove duplicated Joints, Members, and/or Elements have been added to the Copy Model, Refine Finite Element, and Extrude Model dialogs. The user can select what to remove by checking the appropriate boxes. An example of one of the updated dialogs is shown below.



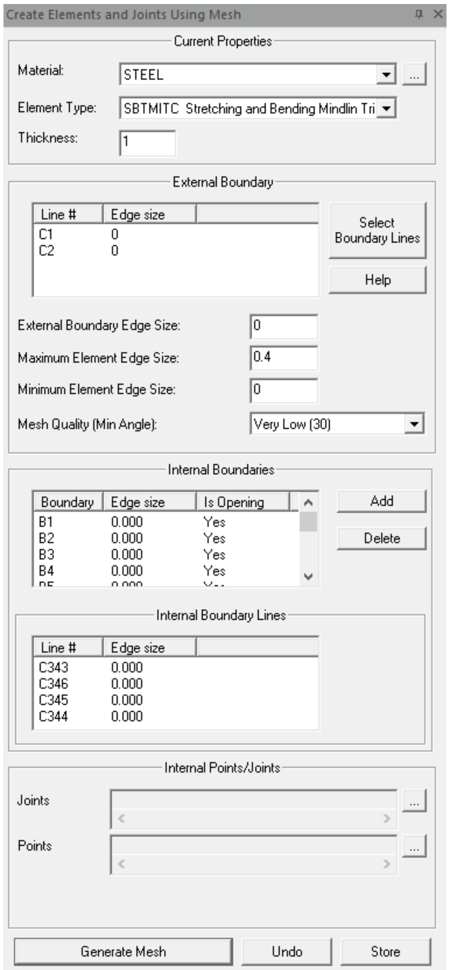
12. A new dialog, “Joints and Elements using Mesh”, was added to help the generation of planar Meshes of Joints and Triangular Elements. Through the dialog, the user can define the parameters required to perform the triangulation.

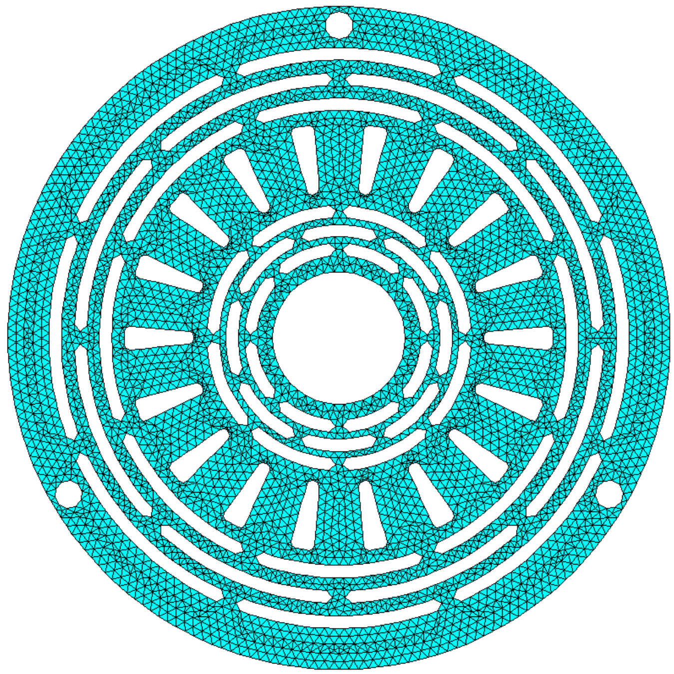
The External boundary as well as the internal boundaries are defined using line/curve objects. The feature gives the user the flexibility to define the maximum element edge size for the entire mesh, for each boundary or for each curve.

The user can also define fix points or joints for the triangulation.

Additionally, the user can set parameter for Mesh quality based on Minimum Angle, and minimum Element Edge size.

The dialog is opened using the menu: Create → Joints and Elements using Mesh.





13. GTMenu View data will now be included in generated input files when the whole model is used for input generation. The View data will be in command format. When View data is input through commands, the external View file, named 'GTMenu_Views.txt' by default, will not be read. The ability to read and write external View files still exists for the occasions that feature is useful.

    This change allows you maintain model and view data within a single file, making it easier to send models for review. Note that only user created views will be included in this data – views #1 to #4 are program defined and cannot be changed by the user.

2.6 CAD Modeler

1. CAD Modeler now supports BricsCAD version 21 including all new functionality.

2. CAD Modeler now supports AutoCAD version 2021 and 2022 including all new functionality.

3. The 2D Area Meshing functionality has been updated to use the same triangulation algorithm used in GTMenu. This algorithm is more stable and produces meshes better adapted to the geometry of the boundaries. This also includes the update of the "Select Mesh Properties" dialog to include the parameters required by the new algorithm.

2.7 Dynamic Analysis

1. The DYNAMIC ANALYSIS NONLINEAR command now supports nonlinearity of the type defined by members having nonlinear spring plastic hinges. This nonlinear condition is defined by the specification of NLS Connection Specs for the desired members under the NONLINEAR EFFECTS Command, illustrated by the following command example:

```
NONLINEAR EFFECTS
    GEOMETRY 1 TO 10
    NLS CONNECTION MEMBERS 1 TO 10 START NLS 'W-PHinge'
```

where 'W-PHinge' is the name of a nonlinear spring curve property defined by the NONLINEAR SPRING PROPERTIES command. This example further illustrates that the NLS CONNECTION nonlinear effect may be specified with other nonlinear effects, GEOMETRY in this case.

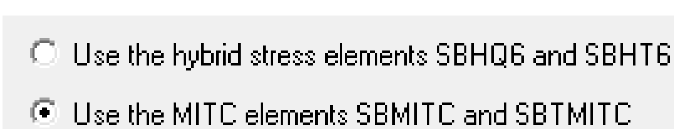
Documentation

The NONLINEAR SPRING PROPERTIES Command, Section 2.5.3.1.3, Volume 3, GT STRUDL User Reference Manual.

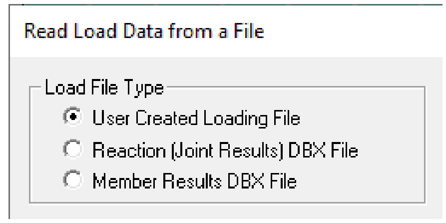
The NONLINEAR EFFECTS Command (NLS Connection Specs), Section 2.5.2.1, Volume 3, GT STRUDL User Reference Manual.

2.8 Base Plate Wizard

1. The MITC elements SBMITC (quadrilateral elements) and SBTMITC (triangular elements) have been added to the Base Plate Wizard as an option to the default hybrid elements SBHQ6 (quadrilaterals) and SBHT6 (triangles). This option can be set on the Plate tab:



2. The Loadings tab now supports reading User Created loading data with three value separators: blank, tab and comma. Originally, only blanks were supported as separators.



Examples:

```
$ -----------LOAD VALUES ---------------
701 1 1 -6331.177 7295.503 261.398 -1071.072 -25524.48 26418.43 $ Blank
701     11      1       -6331.177       7295.503        261.398 -1071.072   -25524.48  26418.43 $ Tab
701,21,1,-6331.177,7295.503,261.398,-1071.072,-25524.48,26418.43 $ Comma
```

2.9 Reinforced Concrete Design

1. New ACI deign codes

A new concrete design codes, ACI 318-08, ACI 318-11, ACI 318-14, ACI 318-19, has been implemented as a released feature. Ultimate strength design (USD) can be used for member design. Implemented ACI design codes (ACI 318-08 to 19) are available for design only and that new design codes can be used only in the design of the columns and rectangular beams.

The user may select either the ASTM (American Standard for Testing and Materials), Canadian Standard, UNESCO, or Korean Standard table of steel reinforcing bar data to design members with implemented ACI design codes. Seismic specifications are not applicable to implemented ACI design codes (ACI 318-08 to 19).

The documentation for the ACI design codes may be found by selecting Help and then Reference Documentation, Reference Manuals, and Volume 4: Reinforced Concrete Design in the GT STRUDL Output Windows.

The METHOD command is used to specify implemented ACI design codes as a design method and select a table of steel reinforcing bar data. New form of METHOD command is given below.

General form:

```
                                                  { ACI318-19       }
                                                  { ACI318-14       }
                                                  { ACI318-11       }
                                                  { ACI318-08       }
                                                  { ACI318-05       }
METHOD ( { → ULTIMATE (STRENGTH) } ) { ACI318-89       } -
         {   WORKING (STRESS)    }   { ACI318-83       }
                                                  { ACI318-77       }
                                                  { ACI318-63       }
                                                  { (BSI) CP110-72  }
                                                  { (BSI) BS8110    }

( BARS { → ASTM                 } ) ( { → NONSEISMIC        } )
       {   CANADIAN (STANDARD)  }     {   SEISMIC           }
       {   UNESCO               }     {   MODERATE SEISMIC  }
       {   KOREAN (STANDARD)    }
```

2.10 Interfaces

1. The DXF Reader has been updated to be able to generate 3-point arcs from the following AutoCad entities: Arcs, Circles, and Arc Segments in Polylines. Additionally, the DXF Reader now applies the required transforms to produce the GTMenu entities in the actual location in 3D space. When using the Members/Elements option of the DXF Reader, each curved segment is converted into two straight members connected at the center of the curve.

Chapter 3

Error Corrections

This chapter describes changes that have been made to GT STRUDL to correct errors. These errors may have produced aborts, incorrect results, or restricted use of a feature in previous versions of GT STRUDL. The error corrections are discussed by the primary feature areas of GT STRUDL.

3.1 GT STRUDL Commands

1. The following issue has been reported as "GPRF-2020.02" and the problem has been fixed for this release. The GT STRUDL steel design code check did not work as documented for the parameters N690-LD and 341-LD which are used to specify the loading conditions applicable for checking the ANSI/AISC N690 nuclear and ANSI/AISC 341 seismic code provisions, respectively. If these Parameters were used, the N690 or 341 provisions may be applied for members or loads where not intended. This issue applied to both AISC14 and AISC15 Design codes.

2. Changing member length due to a structure configuration change will no longer affect the automatic joint type classification for APIWSD21.

3.2 GTShell

1. The Load List Dialog was handling loads incorrectly when their names include a space. i.e. "LD 1". This problem has been fixed for the dialog which is used in GTShell and GTMenu.

3.3 GTMenu

(*GPRF's are not issued for GTMenu unless specifically noted below*)

1. In GT STRUDL 2020, inactive members were ignored and not editable in Member datasheet on GTMenu. The error has been corrected.

2. In GT STRUDL 2020, after splitting members with eccentricities at Place Member Dialog, member incidences were not properly displayed in Member datasheet. The error has been corrected.

3. When opening Model Joint, Member, and Element datasheets, it was required to close dialogs to avoid possible errors. The error has been corrected.

4. When the names of member sections are greater than 24 characters, Member datasheet was aborted. The error has been corrected.

5. Hit Mode option at Create/Edit Note dialog did not work consistently. The error has been corrected.

6. After selecting Joint, Member, or Element Loads, the Area Load wizard button at Edit Load dialog was displayed. The wizard button needs to be displayed only for Area Load at Edit Load dialog. The error has been corrected.

7. After removing all the Physical Member Ids at Create/Edit Group dialog, the Physical Member needs to be deleted but still shows on the list. The error has been corrected.

8. Although there were still Member ids after removing Joint Ids at Create/Edit Group dialog, the group was deleted. The group needs to be deleted only after removing every Id. The error has been corrected.

9. After removing Joint, Member, Element Ids at Create/Edit Group dialog, Inquire Current option was not working properly. The error has been corrected.

10. After adding and removing Joint, Member, Element Ids at Create/Edit Group dialog, type of groups was not properly updated. Individual and mixed group need to be updated as general type.

11. There was a bug in filtering member status from Member datasheet. The error has been corrected.

12. The Area Load Dialog is ignoring the new Joints and Members created after the GTMenu session is initiated. This problem has been fixed and now all existing and new Joints and Members can be used to define Area loads.

13. GT Menu translation and Results viewing is slow and may even abort for a large model with many loads. The reported model of an offshore platform has 28000 joints, 7700 members, 27000 elements and 2100 loads. Analysis times are good, but translation into GTMenu takes almost 2 minutes (almost all load processing), even when GTMENU NO LOADS is used. Results → Diagrams is unresponsive or aborts. This problem has been fixed and now GTMenu will load the results data when needed instead of at initialization.

14. The Help button on the Generate Input Dialog is not working. This problem has been fixed.

3.4 Import CAESAR II Pipe Loads

1. The profile mapping function (read CIS/2 profile name and map to GT STRUDL Table and Section) is case-sensitive and can cause problems when the two systems use 'X' vs 'x' for size separation, e.g. 'W14x54' vs 'W14X54'. This problem has been fixed.

3.5 CAD Modeler

1. In machines with no previous versions to GT STRUDL 2020, CADModeler fails to launch. This condition is due to CADModeler dependency to the “working directory history registry”. This problem has been corrected.

Chapter 4

Known Deficiencies

This chapter describes known problems or deficiencies in Version 40. These deficiencies have been evaluated and based on our experience, they are seldom encountered or there are workarounds. The following sections describe the known problems or deficiencies by functional area.

4.1 CAD Modeler

*(GPRF's are **not** issued for CAD Modeler unless specifically noted below)*

1. Loads are not copied or mirrored when using the Copy or Mirror commands.

2. The Beta angles and Loads are not rotated or mirrored when using the Rotate or Mirror commands.

4.2 Finite Elements

1. The ELEMENT LOAD command documentation indicates that header information such as type and load specs are allowed. If information is given in the header and an attempt is made to override the header information, a message is output indicating an invalid command or incorrect information is stored. (GPRF 90.06)

4.3 General Input/Output

1. Numerical precision problems will occur if joint coordinate values are specified in the JOINT COORDINATES command with more than a total of seven digits. Similar precision problems will occur for joint coordinate data specified in automatic generation commands. (GPRF 2000.16)
2. Internal member results will be incorrect when all of the following conditions are present:
    1. Dynamic analysis is performed (response spectra or time history)
    2. Pseudo Static Loadings are created
    3. Buckling Analysis is Performed
    4. Internal member results are output or used in a subsequent steel design after the Buckling Analysis. In addition, the eigenvalues and eigenvectors from the

Dynamic Analysis are overwritten by the eigenvalues and eigenvectors from the Buckling Analysis.

We consider this problem to be very rare since we had never encountered a job which contained both a Dynamic Analysis and a Buckling Analysis prior to this error report.

Workaround:

Execute the Buckling Analysis in a separate run which does not contain a dynamic analysis.

Alternatively, execute the Buckling Analysis before the Dynamic Analysis and output the Buckling results and then perform a Dynamic Analysis. The Dynamic Analysis results will then overwrite the buckling multiplier and mode shape which is acceptable since the buckling results have been output and are not used in any subsequent calculations in GT STRUDL.

(GPRF 2004.14)

4.4 GTMenu

*(GPRF's are **not** issued for GTMenu unless specifically noted below)*

1. Gravity loads and Self-Weight loads are generated incorrectly for the TRANS3D element.

   Workaround: Specify the self-weight using Body Forces under Element Loads. ELEMENT LOADS command is described in Section 2.3.5.4.1 of Volume 3 of the GT STRUDL Reference Manual.

   (GPRF 95.18)

2. The Copy Model feature under Edit in the Menu Bar will generate an incorrect model if the model contains the TRANS3D element.

   Workaround: Use the DEFINE OBJECT and COPY OBJECT commands in Command Mode as described in Section 2.1.6.7.1. and 2.1.6.7.5 of Volume 1 of the GT STRUDL Reference Manual.

   (GPRF 95.21)

3. Projected element loads will be displayed incorrectly when they are created or when they are displayed using Display Model → Loads.

   Workaround: Verify that the loads are correct in the GT STRUDL Output Window using the PRINT LOAD DATA command or by checking the reactions using

LIST SUM REACTIONS.

(No GPRF issued)

4. GTMenu is limited to 1,000 views. If more than 1,000 views are created, incorrect displays may occur.
(No GPRF issued)

5. The Deformed Structure display with the Deform between Joints option may produce inconsistent results for nonlinear geometric frame members. The deformed structure may show a discontinuity at the joints.
(No GPRF issued)

6. GTMenu is limited to 10,000 Member Property Groups. If more than 10,000 property groups are created, incorrect results may occur. We have never encountered a model with more than 10,000 property groups.
(No GPRF issued)

7. The Label Structural Attributes options in the Label Settings dialog will not display if the Inquire Output dialog is open. For instance, if you have checked the Support Status option in Label Structural Attributes, the legend for the support status will disappear if the Inquire Output dialog is open.

8. When using the new Read Input File function in GTMenu, the user should check the input file (.gti file) before reading into GTMenu. In some instances, an abort could occur. At a minimum, the user should check for duplicate data such as joints, members, elements, and loadings as well as other data that could conflict with existing data already in the model in GTMenu.

Chapter 5

Prerelease Features

5.1 Introduction

This chapter describes new features that have been added to GT STRUDL but are classified as prerelease features due to one or more of the following reasons:

1. The feature has undergone only limited testing. This limited testing produced satisfactory results. However, more extensive testing is required before the feature will be included as a released feature and documented in the GT STRUDL User Reference Manual.
2. The command formats may change in response to user feedback.
3. The functionality of the feature may be enhanced in response to user feedback.

The Prerelease features are subdivided into Design, Analysis, and General categories. The features in these categories are shown below:

We encourage the user to experiment with these prerelease features and provide us with suggestions to improve these features as well as other GT STRUDL capabilities.

5.2 Design Prerelease Features

5.2.1 A new national annex parameter for EC3-2005 steel design code

A new national annex parameter, “Annex”, has been added to the EC3-2005 steel design code. A country name from Table 1.3-7 may be specified which indicates that the national annex of the specified country to be used for the code check or design. Parameter “Annex” is defined in the Table 1.3-1 and the country names are shown in the Table 1.3-7.

Table 1.3-1

EC3-2005 Code Parameters

Parameter Name	Default Value	Meaning
Annex	EC3	Parameter to specify a national annex country name which is used to automatically set the national annex parameters (e.g., GM0 (γ_{M0}), GM1 (γ_{M1}), GM2 (γ_{M2}), Beta (β), and LamdaLT0 ($\bar{\lambda}_{LT,0}$)). The default value of ‘EC3’ for this parameter indicates that the default values shown for national annex parameters GM0, GM1, GM2, Beta, and LamdaLT0 should be used. An alternative country name will reset national annex parameters to the specified country’s national standards. The country names and the parameter values associated to the specified countries are shown in Table 1.3-7. The country names that are not listed in Table 1.3-7 are accepted but a warning message is given that the EC3-2005 default values are used.

Table 1.3-7

Country Names and the National Annex Parameter Values

Country[1]	National Annex Parameter Values
EC3-2005 (defaults)	GM0 = 1.0, GM1 = 1.0, GM2 = 1.25 Beta = 0.75, LamdaLT0 = 0.4
Cyprus, **Greece**, **Netherlands**[2], **Slovenia**, **Spain**, and **Sweden** use above EC3-2005 default values	
Belgium	For welded cross-sections, Parameter SECTYPE = WELDED Beta = 1.0, LamdaLT0 = 0.2
Bulgaria	GM0 = 1.05, GM1 = 1.05
Denmark	GM0 = 1.1, GM1 = 1.2, GM2 = 1.35
Finland	For welded cross-sections, Parameter SECTYPE = WELDED Beta = 1.0, LamdaLT0 = 0.2
France	For welded cross-sections, Parameter SECTYPE = WELDED Beta = 1.0, LamdaLT0 = 0.2
Germany	GM1 = 1.1
Italy	GM0 = 1.05, GM1 = 1.05 Also see Table 1.3-8 for lateral torsional buckling curve for cross-sections using equation (6.57) of the 1993-1-1:2005(E)
Malaysia	GM2 = 1.1 For welded cross-sections, Parameter SECTYPE = WELDED Beta = 1.0, LamdaLT0 = 0.2
Norway	GM0 = 1.05, GM1 = 1.05
Poland	GM2 = 0.9(f_u / f_y) 1.1

Note: National annex parameters with different values from the EC3-2005 defaults are shown in Table 1.3-7 for each country.

1 The country names that are not listed in Table 1.3-7 are accepted but a warning message is given that the EC3-2005 default values are used.

2 Country names more than 8 characters are stored and displayed based on the first 8 characters.

Table 1.3-7 (continued)

Country Names and
the National Annex Parameter Values

Country[1]	National Annex Parameter Values
EC3-2005 (defaults)	GM0 = 1.0, GM1 = 1.0, GM2 = 1.25 Beta = 0.75, LamdaLT0 = 0.4
Portugal	Beta = 1.0, LamdaLT0 = 0.2
Singapore[2]	GM2 = 1.1 For welded cross-sections, Parameter SECTYPE = WELDED Beta = 1.0, LamdaLT0 = 0.2
UK (United Kingdom)	GM2 = 1.1 For welded cross-sections, Parameter SECTYPE = WELDED Beta = 1.0, LamdaLT0 = 0.2 Also see Table 1.3-9 for lateral torsional buckling curve for cross-sections using equation (6.57) of the 1993-1-1:2005(E)

Note: National annex parameters with different values from the EC3-2005 defaults are shown in Table 1.3-7 for each country.

1 the country names that are not listed in Table 1.3-7 are accepted but a warning message is given that the EC3-2005 default values are used.

2 Country names more than 8 characters are stored and displayed based on the first 8 characters.

Table 1.3-8

Lateral torsional buckling curve for cross-sections using equation (6.57) of the 1993-1-1:2005(E) Italy

Cross-section	Limits	Buckling curve	
Rolled I cross-sections	h/b ≤ 2	b	0.34
	h/b > 2	c	0.49
Welded I cross-sections	h/b ≤ 2	c	0.49
	h/b > 2	d	0.76
For all other cross-sections		d	0.76

Table 1.3-9

Lateral torsional buckling curve for cross-sections using equation (6.57) of the 1993-1-1:2005(E) UK (United Kingdom)

Cross-section	Limits	Buckling curve	
Rolled doubly symmetric I and H sections and hot-finished hollow sections	h/b ≤ 2	b	0.34
	2 < h/b ≤ 3.1	c	0.49
	h/b > 3.1	d	0.76
Angles (for moments in the major principal plane)		d	0.76
All other hot-rolled sections		d	0.76
Welded doubly symmetric sections and cold-formed hollow sections	h/b ≤ 2	c	0.49
	h/b > 2	d	0.76

5.2.2 Design of Flat Plates Based on the Results of Finite Element Analysis (The DESIGN SLAB Command)

The goal of the DESIGN SLAB command is to select reinforcing steel for concrete flat plate systems using finite elements as a tool for the determination of design moments.

Instead of dealing with results on an element-by-element basis, the user will be able to design the reinforcing steel for slab systems based on cuts. Here, the term cut refers to the cross-section of a strip at a particular location to be designed. A cut is defined by two nodes identifying the start and end of the cut, and by an element in the plane of the cut.

Once the definition of the cut has been determined, the resultant forces along the cut are computed using either moment resultants (otherwise known as the Wood and Armer method) or element force results (using the CALCULATE RESULTANT command, as described in Section 2.3.7.3 of Volume 3 of the Reference Manuals). The final design moment is determined by computing the resultant moment acting on the cut for each loading condition and reducing these moments to a design envelope.

Once the design envelope is computed, the cross-section is designed according to ACI 318-05 either using default design parameter or with certain user specified design parameters such as the bar size or spacing.

An important distinction is to note that each cut is designed independently from all other cuts. That is, a cut specified in one region is independent with respect to a design in another region. As such, if the user wishes to use the same bar size over multiple adjacent cuts, this information must be specified for each cut.

The form of the command is as follows:

DESIGN SLAB (REINFORCEMENT) (USING) -

$$\left\{ \begin{array}{l} \text{WOOD (AND) (ARMER)} \left\{ \begin{array}{l} \rightarrow\text{AVERAGE} \\ \text{MAXIMUM} \end{array} \right\} \\ \\ \text{CALCULATE (RESULTANT) (ELEMENT) (FORCES)} \end{array} \right\} \text{(ALONG) -}$$

$$\text{(CUT} \left\{ \begin{array}{l} \text{'a'} \\ i_1 \end{array} \right\} \text{)} \left\{ \begin{array}{l} \text{JOINTS} \\ \text{NODES} \end{array} \right\} \text{list}_1 \text{ ELEMENT list}_2 \text{ (TABLE} \left\{ \begin{array}{l} \rightarrow\text{ASTM} \\ \text{UNESCO} \end{array} \right\} \text{) -}$$

$$* \left\{ \begin{array}{l} \text{TOP (FACE) (BARS } i_2 \text{) (SPACING } v_1 \text{)} \\ \text{BOTTOM (FACE) (BARS } i_3 \text{) (SPACING } v_2 \text{)} \\ \text{BOTH (FACE) (BARS } i_4 \text{) (SPACING } v_3 \text{)} \end{array} \right\} \text{-}$$

$$\left\{ \begin{array}{l} \rightarrow\text{INNER (LAYER)} \\ \text{OUTER (LAYER)} \end{array} \right\} \text{(COVER } v_4 \text{) (LINEAR (TOLERANCE) } v_5 \text{) -}$$

(TORSIONAL (MOMENT) (WARNING) v_6)

where,

'a' or i_1 refer to an optional alphanumeric or integer cut name

list_1	=	list containing ID's of the start and end node of the cut
list_2	=	list containing the ID of an element in the plane of the cut
i_2	=	bar size to be used for bars on the top surface of the slab
i_3	=	bar size to be used for bars on the bottom surface of the slab
i_4	=	bar size to be used for both the top and bottom surfaces of the slab
v_1	=	reinforcing bar spacing to be used on the top surface of the slab
v_2	=	reinforcing bar spacing to be used on the bottom surface of the slab
v_3	=	reinforcing bar spacing to be used on both surfaces of the slab
v_4	=	optional user-specified cover distance for reinforcing bars
v_5	=	linear tolerance used in element selection rules for moment computation
v_6	=	optional ratio of torsion to bending moment allowed on the cross-section
TOP	=	element surface with +Z PLANAR coordinate
BOTTOM	=	element surface with -Z PLANAR coordinate

Explanation:

The DESIGN SLAB command allows the user to communicate all data necessary for the reinforcing steel design. This information is processed, and a design is calculated based on the input. The command is designed to provide varying levels of control for the user to make the command as broadly applicable as possible.

The user must first define the cut. A cut is defined by a start and end node ID, and an element ID in the plane of the cut. The user has the option of giving each cut an alphanumeric name for organizational purposes. The purpose of the required element ID is to determine the appropriate plane to design if multiple planes of finite elements intersect along the cut, as defined by the start and end node. An example where this might occur is the intersection of a slab with a shear wall. In this case, a misleading design could be generated if the slab was designed using the forces in the shear wall. The cut definition constitutes all information required to compute the resultant forces acting along the cut.

The total moment acting on a cut cross-section is computed using one of two methods. The use of moment resultants, also known as the Wood and Armer method, is implemented as the default method. In this method, the moment resultants MXX, MYY, and MXY are resolved on a per node basis along the cut, and either the average effect or the maximum effect on the cut is applied to the entire cross-section.

The other option for moment computation is based on the use of element forces. In this method, the total resultant moment acting on the cross-section is computed using the CALCULATE RESULTANT command, and the element force nodal moments are resolved for each node of each element adjacent to the cut.

Once the cut has been defined, the user may indicate parameters to be used to design the system. The user may constrain the bar size or spacing to a certain value, either for the top face, bottom face, or for both faces. In this case, the final design will utilize the information provided. If the bar size is constrained, the appropriate spacing of bars is determined. If the bar spacing is constrained, the appropriate bar size is determined. In the case that the user supplies a bar size and spacing for the cut, the application will simply check the strength of the cross-section against the computed design envelope according to ACI 318. If the user specifies no design constraints, the application assumes a bar size and designs the section to satisfy ACI 318. As such, the user maintains explicit control over the function of the application.

The user may also specify which layer of bars to be designed, using the modifier INNER or OUTER. These refer to the location of reinforcing bars on each surface. At most slab locations, reinforcement is placed in two perpendicular directions on both surfaces of the slab. Since each layer of reinforcement cannot occupy the same space, one layer must be placed on top of the other. OUTER refers to the layer closest to the surface, while INNER refers to the layer nearest the center of the slab.

All user-specified constraints, such as concrete compressive strength, yield strength, cover, and spacing are checked against ACI minimum/maximum values, as specified in ACI 318-02. The thickness of the cross-section is determined internally based on the modeled thickness of the user-specified element.

With respect to the interpretation of results, "top" always refers to the face of the slab on the +Z PLANAR side of the element, and "bottom" always refers to the face of the slab on the -Z PLANAR side of the element. "Positive bending" refers to bending that produces tension on the bottom face of the slab and compression on the top face, as defined previously. "Negative bending" produces tension on the top face and compression on the bottom face, as defined previously.

Requirements:

The MATERIAL REINFORCED CONCRETE command must be specified before the DESIGN SLAB. The MATERIAL REINFORCED CONCRETE command initializes the RC capabilities of GT STRUDL and sets the relevant material and design quantities to their default values for design. At this point, the user can issue the CONSTANTS command to modify any material properties to be used in the design. The default values are:

ECU	=	0.003
ES	=	29,000,000 psi
FCP	=	4000 psi
FY	=	60,000 psi
PHIFL	=	0.9

The STIFFNESS command must be issued prior to the DESIGN SLAB command. The STIFFNESS command solves the global equilibrium equation and computes the quantities required for the determination of the bending moments that the DESIGN SLAB command uses.

Only elements known to appropriately model the behavior of slab systems are included in the computation of design forces. For a flat plate system, only plate bending, and plate elements are used. Thus, if the user models the system using plane stress / plane strain elements, and then issues the DESIGN SLAB command, a warning message is output, and the command is ignored.

Plate bending elements supported include the BPHT, BPR, BPHQ, CPT, and IPBQQ finite elements. General plate elements supported include the SBCT, SBCR, SBHQ, SBHQCSH, SBHT, SBHT6, and SBHQ6 finite elements.

Usage:

Studies have shown that the CALCULATE RESULTANT ELEMENT FORCE option of the DESIGN SLAB command is only applicable in regions where the cut orientation is generally orthogonal to the directions of principle bending. If the geometry of a region dictates that a cut be oriented non-orthogonally to the principal bending directions, a significant torsional effect may occur. In this case, the Wood and Armer method must be employed due to its ability to correctly compute the ultimate moment in a strong torsion field. In the DESIGN SLAB command, the user is warned if the element force implementation computes a resultant torsion greater than 10% of the resultant bending moment on a particular cross-section. The user may modify the torsion warning threshold via the modifiers TORSIONAL MOMENT WARNING. If there is any question of the orientation of the cut with respect to the directions of principal bending, the user should investigate the behavior in the finite element results section of GTMenu.

Usage Example: Description of Example Structure

The example structure is a rectangular slab system, shown in Figure 5.2.3-1. The clear span of the structure is thirty feet, and the slab strip has a width of ten feet. The two ends of the slab are fully fixed, while the thirty-foot sides are free, resembling a fixed-fixed beam. The slab is one foot thick and constructed of normal strength concrete with FCP = 4000 psi. The example structure can be idealized as a subset of a larger slab system, perhaps the design strip running between two column faces in an interior region. The structure is loaded with a distributed surface pressure of 150 psf over the entire surface of the slab.

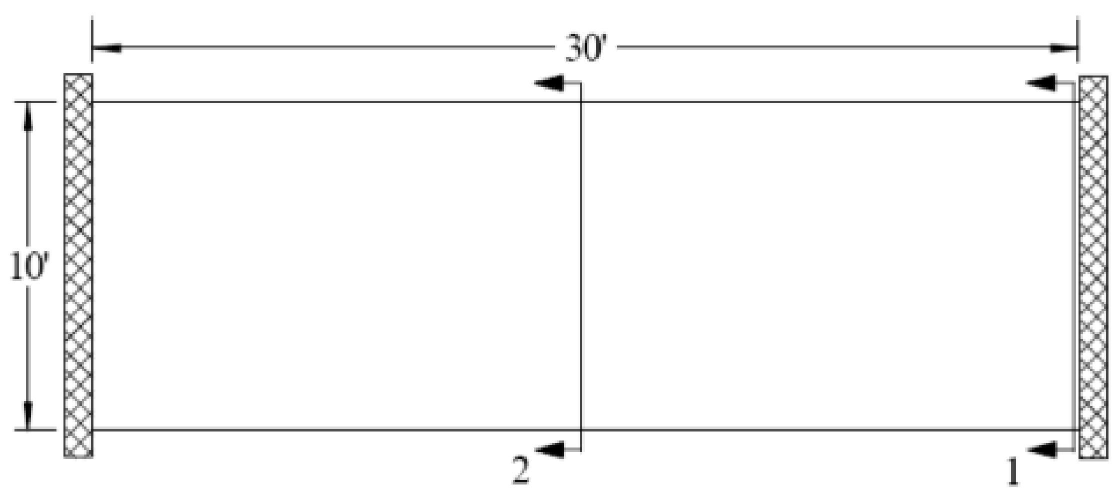


Figure 5.2.3-1 Example Flat Plate Structure (PLAN)

GT STRUDL Finite Element Model

The example structure was modeled in GT STRUDL using PLATE BENDING finite elements. The BPHQ element was utilized, and the configuration modeled corresponded to a mesh of ten elements by thirty elements. The model contained 300 finite elements and 341 nodes. The material properties were the default values associated with the MATERIAL REINFORCED CONCRETE command. All degrees of freedom were restrained at each node along the supported ends of the slab system. Each element was loaded with a surface pressure of 150 psf, resulting in a confirmed summation of vertical reaction of 45,000 lb.

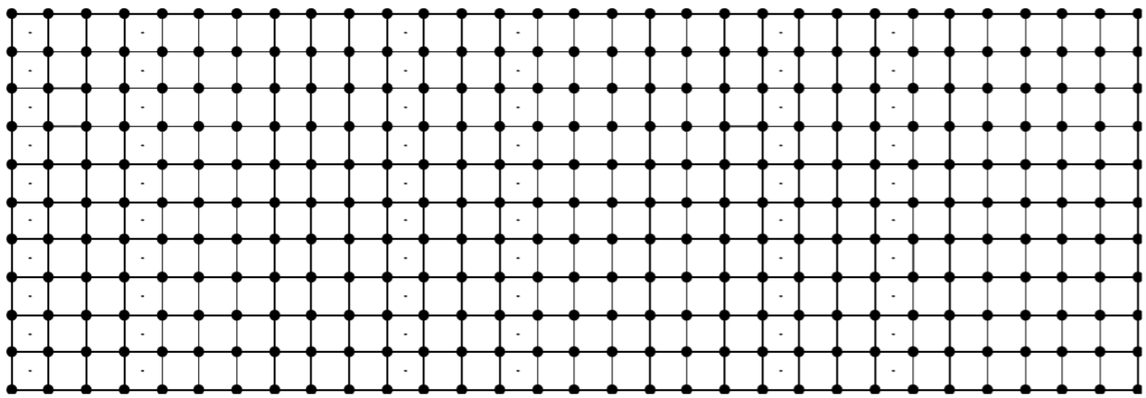


Figure 5.2.3-2 Example Finite Element Model

Definition of Cut Cross-Sections

Two "cuts" are considered for the verification example, as shown in Figure 5.2.3-1.

Cut 1-1:

The cross-section Cut 1-1 is defined along the fixed support at the end of the slab strip and represents the maximum "negative moment" section in the slab where top reinforcing steel would be required. Cut 1-1 originates at node #1 and terminates at node #11. The elements along Cut 1-1 are elements #1-#10. The command given for Cut 1-1 is:

"DESIGN SLAB USING CALCULATE RESULTANT JOI 1 11 ELE 1 TOP BAR 5"

In this case, the user requests that a slab cross-section beginning at node #1, ending at node #11, and in the plane of element #1 be reinforced according to the section moment computed using the CALCULATE RESULTANT command. The user has specified that #5 bars are to be used on the top surface, indicating that spacing is to be computed. The results of the DESIGN SLAB command are shown in the following table.

Calculation	Surface	Bar	Spacing	Area Prov.	Moment Strength	Moment Required
		#	In	sq. in.	lb-in	lb-in
DESIGN SLAB	Top	5	13.0	2.862	1561006.4	1354381.5
DESIGN SLAB	Bottom	NA	NA	NA	NA	NA

The GT STRUDL output for this example is as follows:

```
** FLAT PLATE SLAB DESIGN BASED ON THE RESULTS OF FINITE ELEMENT ANALYSIS **

     PROBLEM - VFE103    TITLE - DESIGN SLAB VERIFICATION - VERIFY DESIGN CALCULATIONS

     RELEVANT ACTIVE UNITS:  INCH LB

     NUMBER OF ACTIVE LOADINGS:     1

     REINFORCEMENT ORIENTATION PERPENDICULAR TO A CUT BEGINNING AT NODE 1
       AND TERMINATING AT NODE 11       AND IN THE PLANE OF ELEMENT 1


** ELEMENT FORCE IMPLEMENTATION **

** DESIGN MOMENT ENVELOPE **

     NEGATIVE MOMENT =      -1354381.48   DUE TO LOAD   150psf
     POSITIVE MOMENT =             0.00   DUE TO LOAD   (none)

     NOTE:
      - Negative moment produces tension on the positive PLANAR Z surface, requiring TOP bars.
      - Positive moment produces compression on the positive PLANAR Z surface, requiring BOTTOM
        bars.


** SLAB CROSS-SECTION **

     Width        Depth          FCP           FY         Cover         Layer
    ________________________________________________________________________

    120.00       12.00        4000.00       60000.00      0.750         Inner


** DESIGN RESULTS (per ACI 318-05) **

     Face        Bar    Spacing   AS PROV'D    MOMENT STRENGTH    MOMENT REQ'D    STATUS
    _________________________________________________________________________________________

     TOP         # 5    13.000      2.862        1561006.4280    1354381.4844     PASSES

     BOTTOM              ( Reinforcement Not Required )
```

Cut 2-2:

The cross-section Cut 2-2 is defined along the center line in the middle region of the slab strip and represents the maximum “positive moment” section in the slab where bottom reinforcing steel would be required. Cut 2-2 originates at node #166 and terminates at node #176. The elements along Cut 2-2 are elements #141-#150 on one side and #151-#160 on the other side. The command given for Cut 2-2 Case 1 is:

“DESIGN SLAB WOOD AND ARMER JOI 166 176 ELE 141 TABLE UNESCO BOTTOM SPACING 10 OUTER LAYER"

In this case, the user requests that a slab cross-section beginning at node #166, ending at node #176, and in the plane of element #141 be reinforced according to the average effect produced by the Wood and Armer method. The user has specified that UNESCO metric reinforcing bars are to be used. The bottom reinforcement spacing has been constrained to 10 inches, and the reinforcement to be designed is located in the outer layer. The results of the DESIGN SLAB command are shown in the following table:

Calculation	Surface	Bar	Spacing	Area Prov.	Moment Strength	Moment Required
		#	in	sq. in.	lb-in	lb-in
DESIGN SLAB	Bottom	M14	10.0	2.864	1664920.7	671358.2
DESIGN SLAB	Top	NA	NA	NA	NA	NA

The GT STRUDL output for this example is as follows:

```
  ** FLAT PLATE SLAB DESIGN BASED ON THE RESULTS OF FINITE ELEMENT ANALYSIS **

       PROBLEM - VFE103    TITLE - DESIGN SLAB VERIFICATION - VERIFY DESIGN CALCULATIONS

       RELEVANT ACTIVE UNITS:  INCH LB

       NUMBER OF ACTIVE LOADINGS:     1

       REINFORCEMENT ORIENTATION PERPENDICULAR TO A CUT BEGINNING AT NODE 166
         AND TERMINATING AT NODE 176      AND IN THE PLANE OF ELEMENT 141


  ** WOOD & ARMER IMPLEMENTATION **

       Design using average result acting on section.


  ** DESIGN MOMENT ENVELOPE **

       NEGATIVE MOMENT =             0.00   DUE TO LOAD   150psf
       POSITIVE MOMENT =        671358.19   DUE TO LOAD   150psf


NOTE:
       - Negative moment produces tension on the positive PLANAR Z surface, requiring TOP bars.
       - Positive moment produces compression on the positive PLANAR Z surface, requiring BOTTOM
bars.


  ** SLAB CROSS-SECTION **

       Width        Depth          FCP           FY        Cover         Layer
      ________________________________________________________________________

      120.00       12.00        4000.00       60000.00     0.750         Outer

  ** DESIGN RESULTS (per ACI 318-05) **

       Face       Bar    Spacing   AS PROV'D    MOMENT STRENGTH    MOMENT REQ'D    STATUS
      _________________________________________________________________________________________

       TOP                 ( Reinforcement Not Required )

       BOTTOM     M14    10.000      2.864        1664920.7190     671358.1875     PASSES
```

5.2.3 ASCE4805 Code for the Design of Steel Transmission Pole Structures

The steel design code, ASCE4805, which is based on the 2005 edition of the ASCE/SEI, *Design of Steel Transmission Pole Structures* Specification has been implemented as a pre-release feature. The ASCE/SEI 48-05 Specification is based on ultimate strength methods using factored loads.

The ASCE4805 Code may be used to select or check any of the following shapes:

Design for axial force, bi-axial bending, and torsion:

Pipes

Regular Polygonal Tubes

Structural Tubing

The documentation for the ASCE4805 code may be found by selecting the Help menu and then Reference Documentation, Reference Manuals, Steel Design, and “ASCE4805" in the GT STRUDL Output Window.

5.3 Analysis Prerelease Features

5.3.1 The CALCULATE ERROR ESTIMATE Command

The form of the command is as follows:

```
CALCULATE ERROR (ESTIMATE) (BASED ON) -

 * { ENERGY (NORM)                                      }
   { MAX DIFFERENCE                                     }
   { DIFFERENCE FROM AVERAGE                            }
   { PERCENT MAX DIFFERENCE                             } -
   { PERCENT DIFFERENCE FROM AVERAGE                    }
   { NORMALIZED PERCENT MAX DIFFERENCE                  }
   { NORMALIZED PERCENT DIFFERENCE FROM AVERAGE         }

         * { TOP    }
  (AT)     { MIDDLE } (SURFACES) (FOR) { →ALL         }
           { BOTTOM }                  {  ELEMENT list }
```

The results from this command provide an estimate of the errors in the finite element discretization of the problem. Energy norm (L2 norm) and nodal error estimates are available.

The L2 norm is given by:

$$\left\| e_\sigma \right\|_{L2} = \left(\int_\Omega (e_\sigma)^T (e_\sigma)\, d\Omega \right)^{1/2}$$

where e_σ is the error in stress, and Ω is the domain of the element. The error stress is the difference between the average stress, σ^*, and element stress at the nodes, σ. The stress norm is obtained by using the shape functions used for displacements, thus,

$$\left\| e_\sigma \right\|_{L2} = \left(\int_\Omega (\sigma^* - \sigma)^T N^T \bullet N (\sigma^* - \sigma)\, d\Omega \right)^{1/2}$$

where N is the shape functions used for the assumed displacement field of the element. The stress norm uses the average stresses and is given by:

$$\left\| \sigma \right\|_{L2} = \left(\int_\Omega (\sigma^*)^T N^T \bullet N (\sigma^*)\, d\Omega \right)^{1/2}$$

The relative percentage error which is output for each element is given by:

$$\eta = \frac{\|e_\sigma\|}{\|\sigma\| + \|e_\sigma\|} \times 100$$

The nodal error estimates the accuracy of the data in a selected nodal output vector. Six nodal error estimation methods are available:

- Maximum Difference.
- Difference from Average.
- Percent Maximum Difference.
- Percent Difference from Average.
- Normalized Percent Maximum Difference.
- Normalized percent Difference from Average.

These error estimates look at the variations in stresses at the nodes. An error estimate of nodal output data will be based on the gradients that data produces in each element. That is, how the data varies across that node based on the different data values from the elements connected at that node. The calculation of error estimates for nodal output is fairly straightforward, the values at each node connected at an element are simply compared. The six nodal error measures are outlined in more detail below:

Maximum Difference Method

$$\left|\text{Value}_{\text{Max}} - \text{Value}_{\text{Min}}\right|$$

Difference from Average Method

$$\text{MAX}\ \left(\left|\text{Value}_{\text{Max}} - \text{Value}_{\text{Avg}}\right|, \left|\text{Value}_{\text{Min}} - \text{Value}_{\text{Avg}}\right|\right)$$

Percent Maximum Difference Method

$$\left|\frac{\text{Value}_{\text{Max}} - \text{Value}_{\text{Min}}}{\text{Value}_{\text{Avg}}}\right| \times 100\%$$

Percent Difference from Average Method

$$\frac{\text{MAX}\left(\left|\text{Value}_{\text{Max}} - \text{Value}_{\text{Avg}}\right|, \left|\text{Value}_{\text{Min}} - \text{Value}_{\text{Avg}}\right|\right)}{\left|\text{Value}_{\text{Avg}}\right|} \times 100\%$$

Normalized Percent Maximum Difference

$$\left|\frac{\text{Value}_{\text{Max}} - \text{Value}_{\text{Min}}}{\text{Value}_{\text{VectorMax}}}\right| \times 100\%$$

Normalized Percent Difference from Average Method

$$\frac{\text{MAX}\left(\left|\text{Value}_{\text{Max}} - \text{Value}_{\text{Avg}}\right|, \left|\text{Value}_{\text{Min}} - \text{Value}_{\text{Avg}}\right|\right)}{\left|\text{Value}_{\text{VectorMax}}\right|} \times 100\%$$

In each of these calculations, the “Min”, “Max”, and “Avg” values refer to the minimum, maximum, and average output values at the node. The “Vector Max” values refer to the maximum value for all nodes from the individual element stress output vector (maximum value from LIST STRESS output for all nodes). All error estimates are either zero or positive, since all use the absolute value of the various factors.

The choice of an appropriate error estimation method largely depends on the conditions in the model. As many error estimates as required may be calculated. In general, the Max Difference method is good at pointing out the largest gradients in the portions of your model with the largest output values. The Difference from Average Method will also identify areas with the largest output values. In this case however, areas where only one or a few values are significantly different will be accentuated. The Max Difference method will identify the steepest gradients in the most critical portions of your model. The Difference from Average Method will identify just the steepest non-uniform gradients, the ones that vary in only a single direction. The two percentage methods identify the same type of gradients, but do not make any distinction between large and small output values. These methods are to be used only if the magnitude of the output is less important than the changes in output. The two percentage methods estimate the error as a percent of the average stress. However, at nodes where there is a change in sign of the stress, the average stress can become very small and often close to zero. As a result, the value of the error becomes enormous. In order to quantify this error, the error at such nodes is given a value of 1,000 percent. The final two normalized percentage methods are usually the best at quantifying overall errors in area with peak stress values.

The results produced by the CALCULATE ERROR ESTIMATE command may also be contoured in GTMenu. To produce a contour of the error estimate in GTMenu, follow the steps below after performing a STIFFNESS ANALYSIS for a static loading:

1. Enter GTMenu.

2. Select Results, Finite Element Contours, and then Energy & Stress Error Estimates.

3. Select the Estimate Method including Value, Surface, and Stress Component.

4. Select the Loading.

5. Select Display (solid colors or lines) to produce a contour of the error estimate.

6. Select Legend to place a legend on the screen indicating the type of error estimate, loading, and surface.

5.3.2 The CALCULATE ECCENTRIC MEMBER BETA ANGLES Command

General form:

CALCULATE ECCENTRIC (MEMBER) (BETA) (ANGLES) (WITHOUT - COMMAND (LISTING))

Explanation:

Section 1.10.4 states that the member beta angle (the orientation of the member cross section principal axes) is defined with respect to the joint-to-joint position of the member before member eccentricities are applied. However, in certain structural modeling situations it may be more desirable to be able to specify a beta angle value that is defined with respect to the eccentric position of the member, after member eccentricities are applied. To this end, the CALCULATE ECCENTRIC MEMBER BETA ANGLES command has been implemented in order to provide beta angle information that can be used to construct CONSTANTS commands that specify beta angle values that reflect such a need. When issued, the CALCULATE ECCENTRIC MEMBER BETA ANGLES command produces a report that includes the member name, the member's originally-specified or -computed joint-to-joint beta angle value, and an adjusted joint-to-joint beta angle value that if specified, produces a member orientation and associated analysis behavior as if the original beta angle were defined with respect to the eccentric position of the member. The report also includes a listing of CONSTANTS/BETA commands for all affected members that can be easily copied and pasted into a GTSTRUDL command text file. If this command listing is not desired, it can be eliminated by giving the WITHOUT COMMAND LISTING option. An example of the report is reproduced below:

```
{   657} > CALCULATE ECCENTRIC MEMBER BETA ANGLES

**** WARNING_CHKECCBTA -- The CALCULATE ECCENTRIC MEMBER BETA ANGLES command is
                          a prerelease feature.  User feedback and suggestions
                          are welcome.


   ****************************
   *RESULTS OF LATEST ANALYSIS*
   ****************************

   PROBLEM - None

   ACTIVE UNITS  FEET KIP  RAD  DEGF SEC
```

The following report lists adjusted beta angle values that if specified, produce member orientations, including corresponding analysis behavior, as if the ORIGINALLY SPECIFIED beta angles were defined with respect to the eccentric position of the member. This report is for information purposes only. No computational action is taken.

```
                Eccentric Member Beta Angle Check Results
                =========================================
    Member               Original Beta Angle      Adjusted Beta Angle
   --------             ---------------------    ---------------------
   11002                         0.06655                  0.09484
   12002                        -0.02815                  0.00884
   11003                        -3.04469                 -3.06850
   13002                         1.26565                  2.52545
   14002                         1.16144                  2.31630
   15002                         1.05723                  2.10572
   16002                         0.95302                  1.89668
   13003                         1.26565                 -0.61557
   14003                         1.16144                 -0.79819
   15003                         1.05723                 -1.03473
   16003                         0.95302                 -1.24443
   17002                        -0.06191                  0.01547
   18002                        -0.44292                 -0.58340
   18003                         3.13987                  3.35983

 CONSTANTS/BETA Commands for Adjusted Beta Angles
 ================================================

 UNITS RAD
 CONSTANTS
   BETA       0.09484  MEMBER  '11002   '
   BETA       0.00884  MEMBER  '12002   '
   BETA      -3.06850  MEMBER  '11003   '
   BETA       2.52545  MEMBER  '13002   '
   BETA       2.31630  MEMBER  '14002   '
   BETA       2.10572  MEMBER  '15002   '
   BETA       1.89668  MEMBER  '16002   '
   BETA      -0.61557  MEMBER  '13003   '
   BETA      -0.79819  MEMBER  '14003   '
   BETA      -1.03473  MEMBER  '15003   '
   BETA      -1.24443  MEMBER  '16003   '
   BETA       0.01547  MEMBER  '17002   '
   BETA      -0.58340  MEMBER  '18002   '
   BETA       3.35983  MEMBER  '18003   '
```

Note that members are listed only if they are active, they have global eccentricities, and the originally-specified beta angle and the adjusted beta angle differ by more than 1o.

5.4 General Prerelease Features

5.4.1 ROTATE LOAD Command

The ROTATE LOAD command will rotate an existing loading and create a new loading condition in order to model a different orientation of the structure or the loading. The ROTATE command is described below and is numbered as it will appear when added to Volume 1 of the GT STRUDL User Reference Manual.

2.1.11.4.6 The ROTATE LOAD Command

General form:

$$\underline{\text{ROT}}\text{ATE } \underline{\text{LOA}}\text{DING } \left\{ \begin{matrix} i_R \\ 'a_R' \end{matrix} \right\} (\underline{\text{ANG}}\text{LES}) \, [\underline{\text{T1}}] \, r_1 \; [\underline{\text{T2}}] \, r_2 \; [\underline{\text{T3}}] \, r_3$$

Elements:

i_R/'a_R' = integer or alphanumeric name of the existing independent loading condition whose global components are to be rotated.

r_1, r_2, r_3 = values in current angle units of the load component rotation angles θ_1, θ_2, θ_3 as shown in Figure 2.1.7-1, Volume 1, GTSTRUDL User Reference Manual.

Explanation:

In many instances, loading conditions are defined for a structure having a given orientation in space, but then the same structure may need to be analyzed for different additional orientations. Applied loading components that are defined with respect to local member or element coordinate systems remain unchanged regardless of the structure's orientation. However, loading components that are defined with respect to the global coordinate system may need to be rotated in order to properly reflect a new orientation for the structure. This is particularly true for self-weight loads, buoyancy loads, etc.

The ROTATE LOADING command is used to take the global applied loading components from an existing loading condition, rotate them through a set of rotation angles, and copy the new rotated global components to a new or modified different destination loading condition. The existing independent loading condition, the ROTATE load, from which the rotated global load components are computed is specified by the loading name i_R/'a_R'. The angles of rotation are specified by the values r_1, r_2, r_3. These rotation angles are defined according to the same conventions as those that define the local support release directions in the JOINT RELEASE command described in Section 2.1.7.2, Volume 1 of the GT STRUDL User Reference Manual, and illustrated in Figure 2.1.7-1.

The ROTATE LOADING command is always used in conjunction with one of the following loading definition commands: LOADING, DEAD LOAD, and FORM LOAD. These commands will define the name (and title) of a new or existing destination loading condition into which the ROTATE LOADING results are copied. The ROTATE LOADING command may be given with any additional applied loading commands such as JOINT LOADS, MEMBER LOADS, ELEMENT LOADS, etc.

Taking the specified loading i_R/'a_R', the ROTATE LOADING command performs the following operations and copies the results into the destination loading condition:

1. Rotate all joint loads, including applied joint support displacements.

2. Rotate all member force and moment loads defined with respect to the global coordinate system. Member force and moment loads defined with respect to the member local coordinate system are simply copied without rotation.

3. Rotate all element force loads defined with respect to the global coordinate system. Element force loads defined with respect to any applicable local or planar coordinate systems are copied without rotation.

4. All other types of loads such as member temperature loads, member distortions, joint temperatures, etc. are copied without changes.

Examples:

1.
```
UNITS DEGREES
LOADING 2 'ROTATED LOADING'
MEMBER DISTORTIONS
     1 TO 10 UNIFORM FR LA 0.0 LB 1.0 DISPL X 0.001
ROTATE LOADING 1 ANGLES T1 45.0
```

The applied loads from previously defined loading 1 will be processed according to Steps 1 to 4 above and copied into the new destination loading 2, which includes the specified member distortion loads applied to members 1 to 10.

2.
```
UNITS DEGREES
CHANGES
LOADING 3
ADDITIONS
ROTATE LOAD 4 ANGLES T2 -30.0
```

Previously defined loading 3 is specified in CHANGES mode, followed by a return to

ADDITIONS mode. The ROTATE LOAD command is then given to add the components of load 4, including appropriate rotations, to loading 3.

Error Messages:

Incorrect data given in the ROTATE LOADING command will cause the following error conditions to be identified and error messages printed:

1. The following error message is printed if the ROTATE loading name is identical to the name of the destination load. An example of the commands that produce this error are also included:

```
{   114} > LOADING 201
{   115} > ROTATE LOAD 201 T1 45.0

**** ERROR_STROLO -   The ROTATE loading is illegally the same as the destination
                      loading.
                      Command ignored.
```

   Loading 201 is illegally named as both the destination load and the loading whose components are rotated.

2. In the following error example, loading 51 is undefined.

```
{   111} > LOADING 201
{   112} > ROTATE LOAD 51 T1 45.0

**** ERROR_STROLO -   Loading to be rotated undefined.
                      Command ignored.
```

3. The following error message is produced because loading 4, specified as the ROTATE load, is a load combination, or dependent loading condition. The ROTATE load must be an independent loading condition.

```
{   141} > LOADING 108
{   142} > ROTATE LOADING 4 T3 45.0

**** ERROR_STROLO -   Rotated Loading 4 is an illegal dependent load.
                      Command ignored.
```

4. This error condition and message is caused by the fact that the destination load 108 is defined as a loading combination.

```
{   144} > LOAD COMB 108 'ALL' COMBINE 1 1.5 2 1.0 3 1.0
{   145} > ROTATE LOADING  1 T3 45.0

**** ERROR_STROLO -   Destination independent loading not defined.
                      Rotated load components not computed.
```

5.4.2 REFERENCE COORDINATE SYSTEM Command

General form:

$$\underline{\text{REF}}\text{ERENCE (}\underline{\text{COO}}\text{RDINATE) (}\underline{\text{SYS}}\text{TEM)} \begin{Bmatrix} i_1 \\ \text{'}a_1\text{'} \end{Bmatrix} \text{ -}$$

$$\left\{ \begin{matrix} \text{(}\underline{\text{ORI}}\text{GIN [}\underline{\text{X}}\text{]}\ v_x\ \text{[}\underline{\text{Y}}\text{]}\ v_y\ \text{[}\underline{\text{Z}}\text{]}\ v_z\text{) (}\underline{\text{ROT}}\text{ATION [}\underline{\text{R1}}\text{]}\ v_1\ \text{[}\underline{\text{R2}}\text{]}\ v_2\ \text{[}\underline{\text{R3}}\text{]}\ v_3\text{)} \\ \begin{Bmatrix} \underline{\text{JOI}}\text{NT} \begin{Bmatrix} i_2 \\ \text{'}a_2\text{'} \end{Bmatrix} \\ \underline{\text{X}}\ v_4\ \underline{\text{Y}}\ v_5\ \underline{\text{Z}}\ v_6 \end{Bmatrix} \begin{Bmatrix} \underline{\text{JOI}}\text{NT} \begin{Bmatrix} i_3 \\ \text{'}a_3\text{'} \end{Bmatrix} \\ \underline{\text{X}}\ v_7\ \underline{\text{Y}}\ v_8\ \underline{\text{Z}}\ v_9 \end{Bmatrix} \begin{Bmatrix} \underline{\text{JOI}}\text{NT} \begin{Bmatrix} i_4 \\ \text{'}a_4\text{'} \end{Bmatrix} \\ \underline{\text{X}}\ v_{10}\ \underline{\text{Y}}\ v_{11}\ \underline{\text{Z}}\ v_{12} \end{Bmatrix} \end{matrix} \right\}$$

Explanation:

The REFERENCE COORDINATE SYSTEM is a right-handed three-dimensional Cartesian coordinate system. The Reference Coordinate System's origin may be shifted from the origin (X=0.0, Y=0.0, Z=0.0) of the overall global coordinate system. The Reference Coordinate System axes may also be rotated from the corresponding orthogonal axes of the overall global coordinate system.

At the present time, this command is used to specify additional coordinate systems which may be used in GTMenu (see Volume 2 of the GT STRUDL Release Guide) to facilitate the creation of the structural model. Reference Coordinate systems created using the above command will be available as Local systems in GTMenu. In a future release, the user will be able to output results such as joint displacements and reactions in a Reference Coordinate System.

There are two optional means of specifying a Reference Coordinate System:

(1) Define the origin and rotation of coordinate axes of the reference system with respect to the global coordinate system, and

(2) define three joints or the coordinates of three points in space.

In either case, i_1 or 'a_1' is the integer or alphanumeric identifier of the reference coordinate system. For the first option, v_x, v_y, and v_z are the magnitude of translations in active length units of the origin of this system from the origin of the overall global coordinate system. The translations v_x, v_y, and v_z, are measured parallel to the orthogonal axes X, Y, and Z, respectively, of the global system and are positive in the positive directions of these axes; v_1, v_2, and v_3 are the rotation angles R1, R2, and R3 in active angular units between the orthogonal axes of this system and the axes of the overall global coordinate system. The description of these angles is the same as given in Section 2.1.7.2 of Volume 1 of the GT STRUDL User Reference Manuals for rotated joint releases (θ_1, θ_2, and θ_3).

In the second case, three joints are required. Each of the three joints may be defined either by a joint identifier using the JOINT option of the command or by its global X, Y, and Z coordinates. If the joint identifier option is used, however, the coordinates of the joint must be specified previously by the JOINT COORDINATES command. The first time (i_2 or 'a_2' or v_4, v_5, and v_6) defines the origin of the reference system; the X-axis of the reference system is determined by the first and second joints (i_3 or 'a_3' or v_7, v_8, and v_9). The positive X-axis is directed from the first to the second joint. The third joint (i_4 or 'a_4' or v_{10}, v_{11}, and v_{12}) is used to define the XY-plane of the reference system. The positive Y-axis is directed toward the third joint. The Z-axis then is determined by the right-hand rule.

Only one reference system can be specified in one command, but the command may be used any number of times.

Modifications of Reference Systems:

In the changes mode, the translations of the origin and/or the rotations of the axes of the reference system from those of the overall global system can be changed. Only that information supplied in the command is altered. The other data that might be supplied in the command remains unchanged. The CHANGES mode, however, does not work for the second option discussed above (i.e., define a reference coordinate system by three joints or the coordinate of three points in space). The reason is that data for these joints are not stored permanently in GT STRUDL. When this option is used, a reference system is created and its definitions of the system origin, rotation angles, as well as the transformation matrix between the global coordinate system and the reference system are generated and stored as would be for the first option. Therefore, if any of the coordinates for the joints used to specify a reference system is changed after the REFERENCE COORDINATE SYSTEM command has been given, the definition of the reference system remains unchanged. For this reason, care must be taken in using the three joints option in conjunction with the changes of joint coordinates. The reference system should be deleted first if any of the coordinates of the joints used to define the reference system are to be changed. Under the DELETIONS mode, the complete definition of the reference coordinate system is destroyed.

Examples:

a)
```
UNITS DEGREES
REFERENCE COORDINATE SYSTEM 'FLOOR2'   -
   ORIGIN 0.0 15.0 0.0 R1 30.
```

This command creates a Reference Coordinate System called FLOOR2 at Y=15 with the axes rotated 30 degrees about global Z.

b)
```
REF COO 1   -
     X    120   Y   120   Z   -120   -
     X    120   Y   240   Z      0   -
     X   -120   Y   120   Z      0
```

This command creates Reference Coordinate System 1 with its origin at 120, 120, -120 and its X-axis from this origin to 120, 240, 0 and its Y axis is the plane defined by the two previous coordinates and the third coordinate, -120, 120, 0, with the positive Y-axis directed toward the third coordinate.

c)
```
REFERENCE COORDINATE SYSTEM 2   -
    JOINT 10 JOINT 20 JOINT 25
```

This command creates Reference Coordinate System 2 with its origin located at Joint 10 and its X-axis directed from Joint 10 toward Joint 20. The XY plane is defined by Joints 10, 20, and 25 with the positive Y-axis directed toward Joint 25.

d)
```
CHANGES
REFERENCE COORDINATE SYSTEM ‘FLOOR2’   -
    ORIGIN   10   20   30
ADDITIONS
```

The above commands change the origin of the Reference System FLOOR2 defined in a) above. The rotation RI = 30 remains unchanged.

e)
```
DELETIONS
REFERENCE SYSTEM 2
ADDITIONS
```

The above command deletes Reference System 2.

5.4.2-1 Printing Reference Coordinate System Command

General form:

PRINT REFERENCE (COORDINATE) (SYSTEM) $\left\{ \begin{array}{l} \rightarrow \underline{\text{ALL}} \\ \text{list} \end{array} \right\}$

Explanation:

The PRINT REFERENCE COORDINATE SYSTEM command will output the Reference Systems. The origin and rotation angles will be output.

5.4.3 GTMenu SURFACE DEFINITION Command

GTMenu construction geometry commands that are written to an input file have been enhanced with the ability to write/read Surface Definitions. Although this prerelease feature is intended mainly to support the save/restore of Surfaces defined through the GTMenu Graphical Interface, users may be able to edit or create new Surfaces through commands provided the point, curve and surface naming rules are followed.

General Form:

GTMenu SURFACE DEFINITION

$\{ 'a_1' \}$ surface-specs$_1$

•

•

•

$\{ 'a_n' \}$ surface-specs$_n$

Elements:

surface-specs = $\left\{ \begin{array}{l} \text{(PATCH SURFACE SPACING) iu iv patch-specs} \\ \text{(SURFACE OF) REVOLUTION (SPACING) iu iv sor-specs} \end{array} \right\}$

patch-specs = U (CURVES) $'b_1' \ldots 'b_n'$ V (CURVES) $'c_1' \ldots 'c_m'$

sor-specs = (REVOLUTION ANGLE) v axis-specs U (CURVE) $'b_1'$

axis-specs = (AXIS) $\left\{ \begin{array}{l} \text{POINTS } 'd_1' \ \ 'd_2' \\ \text{COORDINATES START } x_1\ y_1\ z_1 \text{ END } x_2\ y_2\ z_2 \end{array} \right\}$

where,

$'a_1', 'a_2', \ldots, 'a_n'$	=	1 to 8 character alphanumeric Surface IDs beginning with S (i.e. S1, S2).
iu, iv	=	integer values representing the number of drawing

segments to use in directions U and V respectively.

‘b_1’, ‘b_2’, ..., ‘b_n’ = 1 to 8 character alphanumeric Line/Curve IDs for U direction. n must be greater than or equal to 1 and less than or equal to 10. Line/Curve IDs begin with C (i.e. C1,C2).

‘c_1’, ‘c_2’, ..., ‘c_m’ = 1 to 8 character alphanumeric Line/Curve IDs for V direction. m must be greater than or equal to 1 and less than or equal to 10. Line/Curve IDs begin with C (i.e. C1,C2).

v = real number representing the angle of revolution.

‘d_1’, ‘d_2’ = 1 to 8 character alphanumeric Point IDs for start and end points of the axis of revolution respectively. Point IDs begin with P (i.e. P1,P2).

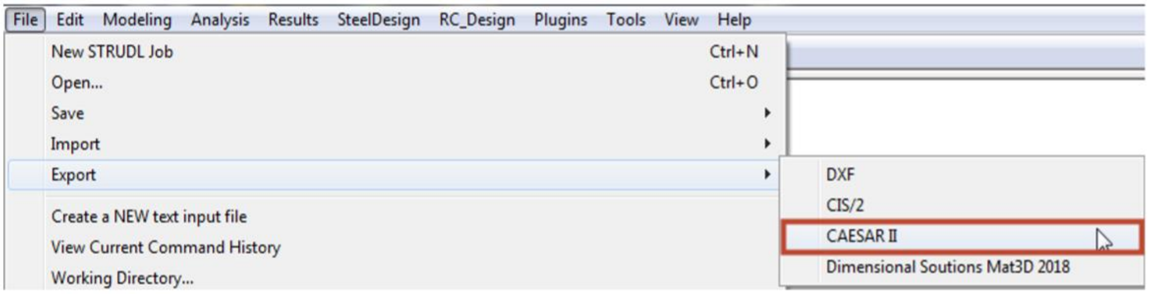
x_i, y_i, z_i = real values representing coordinates for global directions X, Y, Z respectively of the start and end points of the axis of revolution.

Examples:

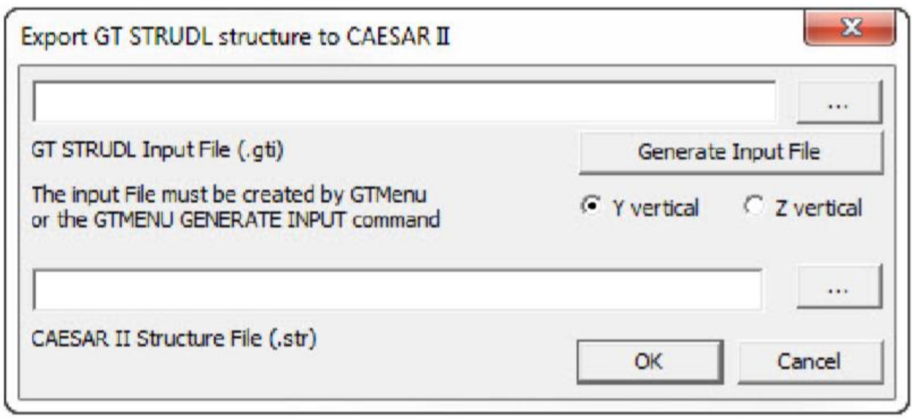
```
GTMenu SURFACE DEFINITION
    'S1' PATCH SURFACE SPACING 10 20 -
        U CURVES 'C1' -
        V CURVES 'C2'

    'S2' SURFACE OF REVOLUTION SPACING 10 20 -
        REVOLUTION ANGLE 60.5 -
        AXIS POINTS 'P1' 'P6' -
        U CURVE 'C2'

    'S3' SURFACE OF REVOLUTION SPACING 10 20 -
        REVOLUTION ANGLE 360 -
        AXIS COORDINATES START  10.0 0.0 10.0 -
                         END    20.0 0.0  0.0 -
        U CURVE 'C2'
```

5.4.4 Export to CAESAR II

You can export the model from GTShell (Command Window) and from CAD Modeler to CAESAR II. In GTShell this feature is available under the File pulldown menu as shown below:

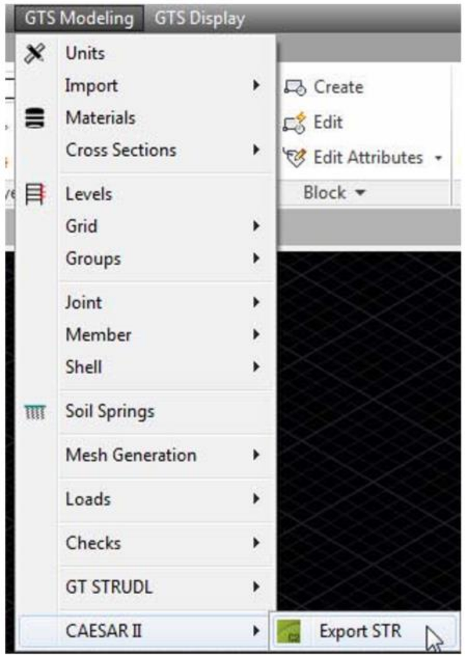


After selecting CAESAR II, the following dialog will pop-up:



This feature will convert a GT STRUDL input file into one or more CAESAR II structure files. The input file must have been created in GTMenu or by using the GTMENU GENERATE INPUT command to ensure a readable (by the translator program) syntax. Note that the GT STRUDL input file must have a ".gti" extension and the specified CAESAR II structural file must have a ".str" extension. You can use the ‘Generate Input File’ button to create a suitable input file from the current GT STRUDL model. Select the appropriate vertical axis (Y or Z) to generate correct Beta angles for the CAESAR II file. Then click the OK button.

In CADModeler, the current drawing can be exported to the CAESAR II Modeler (.str file) from the GTS Modeling pulldown as shown below:



The current drawing can also be exported to into CAESAR II Modeler (.str file) by typing GTSExportSTR at the command prompt. Immediately after the log file appears on the screen. A typical log file is:

```
GT STRUDL Version 2018.R1
GTS2CII Version 2018.R1.01
GTS2CII Binary Dir
C:\\Program Files (x86)\GTStrudl\2018R1\Utilities\GTS2CII\
Project Dir F:\\HexagonPPM\CaesarII\PlantStructure\
Total Number of Sections: 6
Total Number of Joints: 170
Total Number of Members: 233
The model will be saved in 1 STR file(s)
File
F:\\HexagonPPM\CaesarII\PlantStructure\\PStructure_0708_01.str created
```

If the cross sections used in CAD Modeler (and GT STRUDL) are not available in CAESAR's section library, a warning message such as the one shown below will appear:

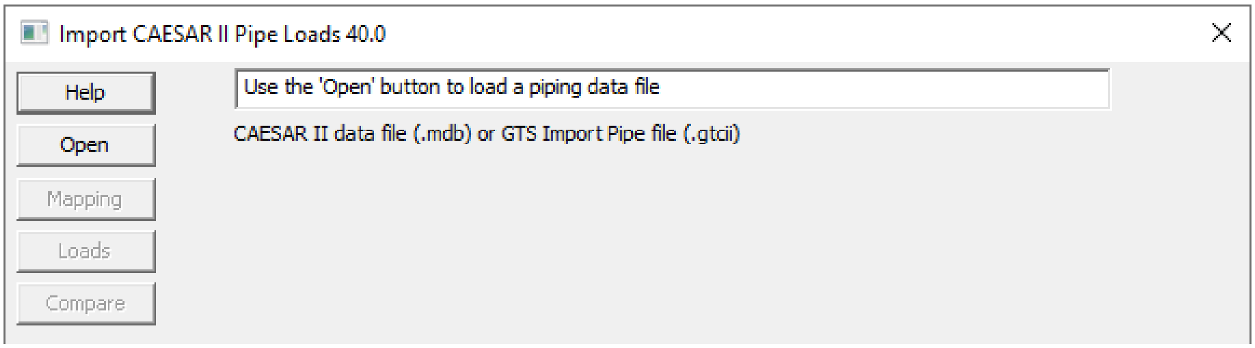
```
WARNING: Section L1x1x1/4 is not available in CII, please use another one
or edit
F:\\HexagonPPM\CaesarII\PlantStructure\\PStructure_0708_01.str file
manually
```

The .str file in CAESAR II is limited to 500 members so if your structure contains more than 500 members, the Export function will automatically break the model into 500 member .str files with a limit of 10 such files (maximum of 5000 members in your structure).

5.4.5 Import CAESAR II Pipe Loads

You can import loads from CAESAR II pipe stress analysis into a GT STRUDL analysis model. Reactions from a CAESAR II .mdb file can be assigned to a GT STRUDL member and applied as a concentrated member load at a designated location. Be sure to export as ".mdb" from CAESAR II. While only a single piping system from each .mdb file can be processed, you can add as many loads from piping systems as needed to a GT STRUDL model by appending the generated loading commands into a single, integrated file for processing by GT STRUDL.

Access to the "Import CAESAR II Pipe Loads" dialog is through the File → Import menu in the GT STRUDL Command window, or through the GTMenu Create → CAESAR II Piping Load Import pick. Click the Help button for information about using the Import CAESAR II Pipe Loads feature.



5.4.6 APPLIED CARDINAL POINTS

A new feature to retain Cardinal Points (point on the section profile that is used to place the member) when importing models from CIS/2 has been added as a Pre-Release feature. The Import CIS/2 process will add the new APPLIED CARDINAL POINTS commands for members that have been designated to be placed by a cardinal point other than the centroid. APPLIED CARDINAL POINT information is retained only for the purpose of exporting CIS/2 files and has no effect on the model geometry.

Since GT STRUDL only uses the centroidal axis to place members, MEMBER ECCENTRICITIES commands are used to correctly position the member upon import. Previously, when the model was exported through CIS/2 all cardinal point information was lost. Now, for members with APPLIED CARDINAL POINTS specified, upon export the original cardinal point is used in the generated CIS/2 .stp file. The eccentricities used to position the member due to cardinal point placement are not included in the generated file, but any specified CIS/2 offsets are retained. This allows programs, such as Smart3D and CADWorx Structure, to create a standard physical model with 'top of steel' or other placement points set correctly from a GT STRUDL CIS/2 file.

Syntax;
APPLIED CARDINAL POINTS
list CP ncp [X] vx [Y] vy [Z] vz
…

Where ncp = cardinal point number. Cardinal points 1 to 10 are accepted.
vx, vy, vz = the <u>global</u> eccentricities due to cardinal point placement

PRINT CARDINAL POINTS (*list*)

Print the existing cardinal point information. If *list* is omitted, the cardinal point information is printed for all members.

DELETIONS
APPLIED CARDINAL POINTS (*list*)
(*list*)
…
ADDITIONS

Delete cardinal point information for members specified in *list*. Note that if you delete the cardinal point information for any member in a Physical Member, all analytical members in the Physical Member will have cardinal point information deleted to maintain Physical Member integrity.